IN-64
14412

P51

Institute for Computational Mechanics in Propulsion (ICOMP)

Fifth Annual Report—1990

May 1991

NASA
LEWIS RESEARCH CENTER

ICOMP

CASE WESTERN
RESERVE UNIVERSITY

NASA
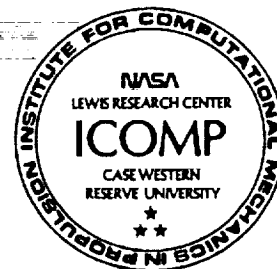
Institute for Computational Mechanics in Propulsion (ICOMP)

Fifth Annual Report—1990

Compiled and edited by
Charles E. Feiler
ICOMP Executive Officer

Approved by
Louis A. Povinelli
ICOMP Director

May 1991

CONTENTS

PAGE ii INTENTIONALLY BLANK

INSTITUTE FOR COMPUTATIONAL MECHANICS

IN PROPULSION (ICOMP)

FIFTH ANNUAL REPORT

1990

SUMMARY

The Institute for Computational Mechanics in Propulsion (ICOMP) is operated jointly by Case Western Reserve University and the NASA Lewis Research Center in Cleveland, Ohio. The purpose of ICOMP is to develop techniques to improve problem-solving capabilities in all aspects of computational mechanics related to propulsion. This report describes the activities at ICOMP during 1990.

INTRODUCTION

The Institute for Computational Mechanics in Propulsion (ICOMP) is jointly operated at the NASA Lewis Research Center by Case Western Reserve University and NASA Lewis Research Center under a Space Act Agreement. ICOMP provides a means for researchers with experience and expertise to spend time in residence at Lewis performing research to improve computational capability in the many broad and interacting disciplines of interest in aerospace propulsion. The organization and operation of ICOMP have been described in ICOMP Report No. 87-8 (NASA TM-100225), "The Institute for Computational Mechanics in Propulsion (ICOMP), First Annual Report," Nov., 1987, 14 pages. The activities of 1987, 1988, and 1989 are described in ICOMP Reports No. 88-1 (NASA TM-100790), No. 89-1 (NASA TM-101961) and No. 90-1 (NASA TM-102519) respectively.

The scope of the ICOMP research program is: to advance the understanding of aerospace propulsion physical phenomena; to improve computer simulation of aerospace propulsion components; and to focus interdisciplinary computational research efforts. The specific areas of interest in computational research include: fluid mechanics for internal flow; structural mechanics and dynamics; and material science.

The report summarizes the activities at ICOMP during 1990. It lists the visiting researchers, their affiliations and time of visit followed by reports of RESEARCH IN PROGRESS, REPORT AND ABSTRACTS, and SEMINARS presented. Two special events during 1990 are also described. These are: (1) the formation of a focus group within ICOMP designated as the Center for Modelling of Turbulence and Transition (CMOTT) and (2) an intensive two week long lecture series entitled, "Instabilities and Transition to Turbulence in Free and Wall Shear Layers," by Mark V. Morkovin, Professor Emeritus at the Illinois Institute of Technology.

THE ICOMP STAFF OF VISITING RESEARCHERS

The composition of the ICOMP staff during 1990 is shown in figure 1. Forty-seven researchers were in residence at Lewis for periods varying from a few days to a year. Figure 2 is a photograph of the ICOMP Steering Committee and the visiting researchers taken at a reception in August 1990. Figure 3 lists the universities or other institutions represented and the number of people from each. The figure lists thirty-six organizations. Figure 4 shows the growth of ICOMP during its first five years in terms of staff size, organizations represented and technical output as measured by the numbers of seminars, reports and workshops. The next sections will describe the technical activities of the visiting researchers starting with reports of RESEARCH IN PROGRESS, followed by REPORTS AND ABSTRACTS, and finally, SEMINARS.

RESEARCH IN PROGRESS

Dare Afolabi, Purdue University

This report covers work done at ICOMP between June and August 1990. My research was motivated by the need to reduce the vibration amplitudes of turbine blades in jet propulsion systems, such as the Space Shuttle Main Engine. Usually, the blades are coupled together, both by the elasticity of the disks on which they are mounted, and the aerodynamics of the working fluid. These kinds of coupling give rise to modal interaction.

Suppose the resonance frequencies of a vibrating system are a function of a number of system parameters (the coupling strength, coupling pattern, mistuning, rotation speed, etc.). As these parameters are varied, several modes of vibration approach one another, leading to modal interaction. When two modes, for example, approach each other, they either repel each other when they get too close, giving rise to the so-called "veering" phenomenon, or they attract each other, giving rise to modal degeneracy. The question of modal degeneracy in dynamic systems, or its impact on system dynamics, is not properly understood at present, although a lot of progress has been made in the recent past. According to Crandall, the problem was first studied by Lagrange, who made an error by assuming the presence of secular terms in his solution for the two degree of freedom system. That Lagrange's error was not detected for several years reminds one that the problems involved are not trivial, although they may appear to be simple.

We have now established that veering takes place in systems with conservative coupling, while degeneracy arises in systems with nonconservative coupling. In future work, the implication of this to the turbine blade vibration problem will be examined, in light of recent mathematical apparatus provided by Arnol'd and others (Refs. 6 and 7).

REFERENCES

Crandall, S.H.; and Afolabi, D.: Further Remarks on Conservative and Nonconservative Coupling in Dynamic Systems. (Forthcoming)
Afolabi, D.; and Nwokah, O.D.I.: On the Dynamics of Nominally Symmetric Systems with Parameter Uncertainties. Lecture Notes in Control and Information Science (Ed.: Thoma), Springer, 1990. (Forthcoming)
Afolabi, D.: Modal Interaction in Dynamic Systems. ICOMP Report, in preparation.
Crandall, S.H.: Private Communication.
Lagrange, J.L.: Analytical Mechanics. Guthier-Villas, Paris (in French).
Arnol'd V.I.: Lectures on Bifurcations and Versal Families. Russ. Math. Surv., vol. 27, no. 5, 1972, pp. 54-123.
Arnol'd, V.I.: Bifurcations and Singularities in Mathematics and Mechanics. Invited Lecture, International Union of Theoretical and Applied Mechanics, 1988 Congress (Elsevier Publishers, 1989).

Suresh K. Aggarwal , The University of Illinois at Chicago

The computational study of turbulent unconfined spray flames was continued. Some minor bugs were fixed in the spray flame code which had been developed during the previous year's ICOMP visit. Results were obtained on the detailed structure of a methanol spray flame formed by the injection of methanol fuel into quiescent air. An important observation is that the turbulent flame structure is both quantitatively and qualitatively different from that of the corresponding gaseous diffusion flame. In addition, the spray flame exhibits a strong sensitivity to the transient liquid-phase processes. The latter is an interesting result since in an earlier study the spray vaporization behavior for the same liquid fuel indicated only a weak sensitivity to these processes. Details will be given in the journal publication currently under preparation.

Kyung H. Ahn, (Postdoctoral), Cleveland State University

My research is concerned with numerical simulation of the oscillating flow in Stirling engine heat exchangers. There has been conflict in determining the effects of the periodic oscillations on the time-averaged properties of the flow. For pulsating flow which has non-zero mean velocity over one cycle, it has

been found that for sufficiently high frequencies, the time-mean properties, (e.g., time-mean velocity profile, time-mean wall shear stress, and power loss due to friction), were affected by imposed unsteadiness and also that a quasi-steady turbulent model does not adequately describe unsteady flow conditions, at least for high frequencies. As a result, a criterion has been defined which attempts to specify the range of a dimensionless parameter for which the assumption of quasi-steady flow is valid. The criterion has been evaluated by comparing experimental results with quasi-steady numerical predictions. This effort seeks to define a similar criterion for which quasi-steady assumptions are valid for the oscillating flow with zero-mean velocity. Computationally, a two-dimensional finite-volume code is being modified to handle oscillating flow and to implement an upgraded k-ϵ turbulence model. Currently, at the University of Minnesota, oscillating flow experiments are being performed under grant from NASA Lewis and the data will be compared with my numerical simulation.

F. A. AKL, Louisiana Tech University

The research has as its objective the determination of the computational characteristics associated with the mapping of the generalized eigensolution of linear elastic finite element models onto a parallel microcomputing architecture. The last phase of developing an auxiliary eigensolver to extract the lowest order eigenpairs has been completed. A parallel eigensolution method using a binary tree transputer network has now been successfully implemented. A PC-based network of 31 transputers configured in a binary tree topology was used to solve for the subspace of the lowest order eigenpairs of the generalized eigenproblem $[K][\Phi] = [M][\Phi][\Lambda]$. Replication of the algorithm on a variable number of processors permits flexibility in tailoring the algorithm to the depth of the binary tree. As originally proposed, the method of recursive doubling (Kogge, 1974) was used in implementing the eigensolver. The theoretical speedup in the number of steps needed to perform a number of associative operations, n, is calculated by Kogge (1974) to be equal to $(n-1)/\log_2 n$. The figure shows typical results obtained for a cantilever beam. The actual behavior of the algorithm is in close agreement with the theoretical prediction given by Kogge (1974) and suggests that increasing the number of processors can significantly enhance the computational speedup.

The results achieved by the method of recursive doubling are very promising in the cases tested during last summer. Transputer networks offer flexibility in implementing an iterative binary subspace eigensolver with an actual computational speedup comparable to theoretical values. Efforts will continue to significantly enhance the capabilities of the algorithm in order to realize the full potential of the new algorithm. This enhancement will include the ability to solve a wide class of problems of interest to NASA and should involve the addition of a library of finite element routines. Further testing, debugging, and verification of the algorithm are needed. The work will be submitted for journal publication and/or conference presentation at appropriate forums soon.

REFERENCES:

Kogge, P.M.: Parallel Solution of Recurrence Problems. IBM J. Res. Dev., vol. 18, no. 2, Mar. 1974, pp. 138-148.

Performance of binary tree eigensolver, number of iterations = 4.
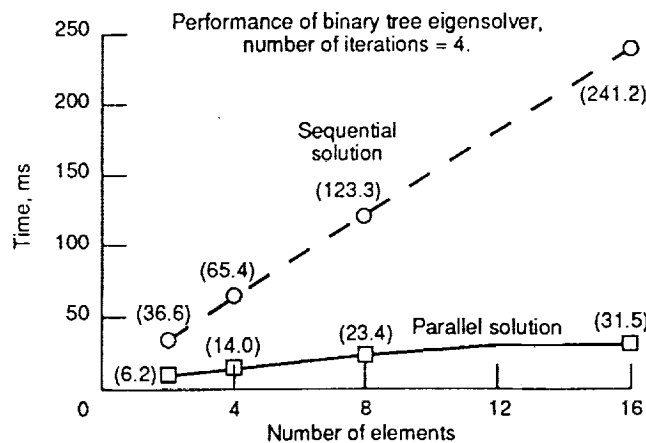
Andrea Arnone, University of Florence, Italy

During my stay at ICOMP, I have continued my work in the field of transonic cascade flow calculations. The final goal of this activity is the development of a fast, robust, and accurate solver for viscous/inviscid flows to be used in turbomachinery design. My activity consists of three parts: (1) generation of grids capable of picking up details of the flow field, (2) computation of 2D flows; and (3) computation of 3D flows.

Rotor and stator cascades of modern turbomachinery are often characterized by a high turning geometry and/or by strong flow deviations from the axial so that the generation of meshes capable of picking up the flow details is not so straightforward. Procedures that use periodicity generally give rise to highly distorted meshes. In the present work, this problem is solved by the introduction of nonperiodic elliptic C-type grids. The removal of periodicity allows the grid to be only slightly distorted even for cascades having a large camber and a high stagger angle. On these new kinds of grids, the Euler and Navier-Stokes equations are efficiently solved using a Runge-Kutta scheme in conjunction with accelerating techniques like multigridding and variable coefficient implicit residual smoothing. The two-layer eddy-viscosity model of Baldwin and Lomax is used for the turbulence closure.

The above procedure has been applied first to 2D flows and later extended to the 3D case. The predicted flows agree well with experiments and both robustness and computational time are very competitive with respect to other existing codes.

Thomas F. Balsa, University of Arizona

At high Mach numbers a supersonic mixing layer is marginally stable at zero frequency (i.e., in the vortex sheet limit) according to a famous result due to Miles based on linear theory. Because of this, a supersonic mixing layer with continuous velocity and temperature profiles is slightly unstable at low frequencies, and the basic objective of this research is to balance the small exponential growth arising from this linear instability against the cumulative effects of small (but non-negligible) nonlinear terms in the equations of motion. This can be done using asymptotic methods and the concepts from nonlinear and viscous critical layer theory. The Navier-Stokes equations are greatly simplified to two coupled nonlinear equations for the perturbation vorticity and temperature (for the case of two-dimensional flow). These equations are solved numerically.

So far, we have studied the inviscid solutions quite extensively. These solutions suggest that the nonlinear terms in the critical layer do not change the growth rate significantly (from that of linear theory); although, they do alter the wave number. Because of this continuous growth of a disturbance, even when the flow is strongly nonlinear, the fluid in the braid region is severely strained. This straining produces large temperature gradients which produce significant vorticity in the braid due to the baroclinic torque. The picture of the roll-up of vorticity is this: the cores of the rollers contain small amounts of vorticity while most of the vorticity (positive and negative) lies in this elongated islands adjacent to the braids.

Preliminary results for the viscous case suggest that even a small amount of diffusion (viscosity and heat conductivity) will reduce the concentration of vorticity in these islands. It is unclear at this point how dissipation affects the amplitude of the disturbance. Work is in progress to clarify the role of dissipation and to identify the next stage of evolution of the disturbance.

Alvin Bayliss, Northwestern University and ELI TURKEL, University of Tel Aviv

We studied the role of adaptively determined mappings in enhancing the accuracy of pseudo-spectral methods in approximating functions which have localized regions of rapid variation. Functions exhibiting such behavior occur in many application area, for example in combustion calculations. A new family of mappings was developed and shown to be extremely effective in enhancing the accuracy of pseudo-spectral methods for such functions. The results of this research were reported in an ICOMP report.

Mappings are generally effective in approximating functions with a single region of rapid variation or with multiple regions which are closely spaced. In order to deal with functions which have widely spaced regions of rapid variation, we consider domain decomposition methods combined with mappings. A study of the role of domain decomposition methods in enhancing the accuracy of pseudo-spectral approximations was undertaken. In particular, we considered the interaction between domain decomposition methods and mappings within the subdomains. We developed procedures which would be suitable for adaptive determination of the location of the subdomains. We demonstrated that the use of mappings, together with

domain decomposition methods could lead to a significant improvement in the approximation of functions with multiple regions of rapid variation. This research is still ongoing.

Richard J. Bodonyi, Ohio State University

During my one-week visit at ICOMP, a research effort concerning the comparison between the classical/numerical and interactive asymptotic theory approaches to the linear and nonlinear boundary-layer stability and transition problems was begun. After a review of the literature, it was decided to concentrate on the 3D aspects of the theory. Furthermore, an interactive model based on the work of Smith, Papegeorgiou, and Elliot (J. Fluid Mech., Vol. 146, 1984) and appropriate for large but finite Reynolds numbers was formulated during the week. Currently, efforts are underway to numerically solve the governing equations for this model. It is hoped that these results will allow us to demonstrate that the different views and opinions on the merit of the alternative methods noted above can be consolidated.

Mark E. Braaten, GE Research & Development Center

During my visit to ICOMP, I worked on implementing a parallel 3D pressure-correction-based Navier-Stokes algorithm on the 128-node iPSC/860 at the NASA Ames Research Center. Before coming to ICOMP, I had developed a preliminary version of the algorithm on a hypercube simulator and tested it briefly on a small machine at Intel Scientific Computers. The algorithm was successfully run on 64 processors of the NAS hypercube. An inviscid test case involving a cascade of biconvex airfoils was run using a 61 x 21 x 21 grid. Preliminary results show a calculation time for 10 iterations of 23 seconds (with 64 processors), which is 34 percent faster than the measured time of 34.8 seconds on a single CrayY-MP processor. Performance on the hypercube is expected to improve substantially with further compiler optimization, further algorithm refinements, and the advent of pipelining compilers for the i860. Overall, the results are extremely encouraging, and demonstrate that current distributed memory MIMD machines offer supercomputer performance for CFD applications.

C. L. Chang, Cleveland State University

I visited ICOMP from May to August 1990 on a part-time basis. During this period, my attention was focused on a least-squares finite element method for the incompressible, 3D Navier-Stokes equation using the velocity-pressure-vorticity formulation. A numerical iteration scheme solving this nonlinear system of PDE's was developed. The mathematical analysis of the convergency and error estimates for the linear Stokes problem and for each nonlinear iteration step have been partially completed. Analysis also shows that the finite element spaces with either a piecewise polynomial or a trilinear "brick" can be chosen to present the velocity u, pressure p and the vorticity ω. No extra restriction is required other than the general requirement for finite element spaces. Error estimates show that this method may achieve optimal rates of convergence in each iteration step. Some preliminary numerical experiments have been done for a 3D driven cavity flow.

Tawit Chitsomboon, (Postdoctoral) Old Dominion University

To support the national aerospace plan project, the RPLUS3D CFD code has been developed at the NASA Lewis Research Center. The code has the capability to solve 3D flow fields with finite rate combustion of hydrogen and air. The combustion processes of the hydrogen-air system are simulated by an 18-reaction path, 8-species chemical kinetic mechanism. The code uses a Lower-Upper (LU) decomposition numerical algorithm as its basis, making it a very efficient and robust code. Except for the Jacobian matrix for the implicit chemistry source terms, there is no inversion of a matrix even though it uses a fully implicit numerical algorithm.

Despite the many good features of the RPLUS3D code, it lacks a turbulence model rendering its range of application very limited. The main purpose of this work is to incorporate a k-epsilon (two equation) turbulence model into the RPLUS3D code. Since January 1990, when this work was started, some of the more important accomplishments are categorized as follows:

1. Add a k-epsilon turbulence model: The model selected was in a high Reynolds number form. The low Reynolds number form could not be used economically in the case of a 3D flow with chemical reaction since it demands too much in computer resources. The addition was designed to be as modular as

possible but some interactions with the mother code are needed in order to be more efficient. The first test case tried was a Mach 0.5 flow over a flat plate. The velocity profile compares very well with the log-law profile. The friction coefficient also compares well with the Van Driest correlation. More validations will be performed for other flows such as free shear layer and jet flows.

2. Improved accuracy and convergence rate: According to a stability analysis of a model equation, it is shown that the RPLUS3D code excessively added artificial damping to the right hand side of the algorithm while at the same time it overestimated the spectral radii on the left hand side. These excessive additions would not give an optimum convergence rate. Modifica-tions were made such that true directional spectral radii were added to the left hand side and the artificial damping terms were reduced to optimum values in accord with the stability analysis. A test run was made of a Mach 4 flow of air over a 10 degree compression ramp. It was found that the modified code converged to machine zero about five times faster than the original code while at the same time it somewhat improved the shock resolution.

3. Added consistent damping terms at block interfaces: it had been observed that at the block interface of a multi-block grid, wiggles developed. This happened because the damping terms at the interface were not consistent with those at the interior points.

4. Validated RPLUS3D with laminar flow over flat plates: The test cases run were for Mach numbers of 0.1, 0.3 and 0.5. Results of all cases seemed to be good except for the region of high curvature of the velocity profile near the edge of the boundary layer.

5. Add two dimensional capability to the code: It turned out that this is not a trivial task especially for a finite volume code like RPLUS3D. It is nice that now the code can solve a 2D flow without having to carry the third direction along as a redundancy. There is, of course, no need to maintain a separate 2D code.

6. Implemented a local time stepping capability: The original code always ran at a fixed time step of one second. For most flows this corresponds to using a very large CFL number which may not be conducive to a fast convergence rate. With the local time stepping, it was found that an optimum CFL number was, in agreement with other investigators, around 5 to7.

7. Implemented implicit boundary conditions: This addition enhanced the convergence rate by about 30 percent at the expense of a more complex code and an increase of about 20 percent in CPU time per iteration step. There seems to be, then, no net advantage of the implicit boundary condition except maybe in the area of robustness.

8. Changed input file: Instead of having to scan a whole subroutine to set up a problem, a user now can change numbers in a small file of length about one page. To start up a run from a previous run one now needs only to change a parameter in this input file without having to recompile the input subroutine as before.

9. Solved two 3D flow fields of W. Hingst and D. Davis: This work was performed in collaboration with Dr. A. C. Taylor of Old Dominion University. My task was to set up the program for these particular problems and to generate the grid for one of the problems.

Stephen Cowley, Imperial College

During my visit to ICOMP, I continued writing a paper on "The Instability of Hypersonic Flow Past a Flat Plate" (co-authors Philip Hall and Nik Blackaby). It is demonstrated in this paper that it is important to use the correct (Sutherland) viscosity law if the instability properties of hypersonic flow are to be correctly understood. We show how the presence of a leading edge shock can significantly change the growth rate of unstable modes, and we identify the crucial role played by the viscous adjustment region that is between the upper inviscid shock-layer and lower high-temperature boundary layer. Numerical Navier-Stokes solutions must be accurate in this region if the instability properties of the flow are to be correctly understood. In addition, we show how the viscous blowing velocity normal to the plate needs to be included in a correct description of the nature of instability modes with wavelengths comparable with the thickness of the boundary layer. This work is of relevance to the design of a hypersonic transport vehicle.

I consulted with David Wundrow and Marvin Goldstein on aspects of the numerical solution of a nonlinear, nonequilibrium, viscous critical layer in hypersonic boundary layers.

Together with Marvin Goldstein and Stewart Leib, I worked on a problem concerning the growth of 3D disturbances in flow past the leading edge of a thin, blunt body. Upstream it was assumed that the uniform flow was slightly disturbed by a vorticity field with a direction normal to the plate; the typical spanwise scale

of the disturbance was assumed to be comparable with the thickness of the body. Sufficiently far downstream, even a very small spanwise disturbance can lead to order one changes in the mean flow due to a disparity in length-scales; in addition, vortex stretching effects complicate the flow. An analytic description of this problem was derived, and numerical results obtained. The destabilization of the boundary layer due to these mean flow effects was highlighted. A paper is in preparation.

Robert J. Deissler, Los Alamos National Lab

In collaboration with Dr. Wai Ming To, I am writing a code to solve the fluid equations for Rayleigh-Benard convection with a through flow. More specifically, a liquid flows horizontally through a pipe with rectangular cross section, and the top and bottom of the pipe are at different temperatures. Experimentally this system has been observed to exhibit very interesting behavior. For example, for a range of fixed Reynolds numbers, as the Rayleigh number is gradually increased, longitudinal rolls eventually appear at the down-stream end of the pipe. For larger Rayleigh number, more of the pipe is filled with the longitudinal rolls, until at sufficiently large Rayleigh number essentially the entire pipe is filled. As the Rayleigh number is further increased, transverse rolls eventually appear at the downstream end replacing the longitudinal rolls. For larger Rayleigh number, a larger fraction of the pipe is filled with the transverse rolls, until for sufficiently large Rayleigh number essentially the entire pipe is filled with transverse rolls.

One of the most interesting questions is the role of external noise in this system. Recently, this system has been modelled with a pair of coupled Ginzberg-Landau equations (H.R. Brand, R.J. Deissler, and G. Ahlers, "A Simple Model for the Benard Instability with Horizontal Flow Near Threshold," submitted for publication). It was found that this simple model qualitatively reproduced the behavior described above and that external noise and noise-sustained structures (R.J. Deissler, J. Stat. Phys. 40 (1985) 371; Physica D25 (1987) 233; J. Stat. Phys. 54 (1989) 1459) played a crucial roll in the behavior.

Here at NASA, Dr. To and I plan on studying this problem by directly solving the 3D fluid equations with inflow and out-flow conditions with noise introduced near the entrance of the pipe. A time-splitting method will be used with the diffusive part being solved exactly. The result will be a highly stable code.

A. O. Demuren, University of Lagos*

Current research work focuses on implementation of second-moment turbulence closure models suitable for application to complex 3D flows. The most promising are the full Reynolds Stress Models (RSM) based on the numerical solution of partial differential equations for the turbulent (Reynolds) stresses and associated turbulence fluxes, and a representative equation for the length scale. In the present work, the model proposed by Launder, Reece, and Rodi (JFM, 1975) has been generalized for curvilinear coordinates and incorporated in a multigrid code which ensures efficient convergence on any grid size. Demuren (1989, 1990a) has shown that the full Reynolds stress equations can be solved with little difficulty and minimal additional cost by a splitting of the source terms. The resulting system is more robust and less sensitive to model coefficients. Computations show similar level of agreement with experimental data in wall-bounded flows, but better agreement in nonwall bounded flows (Demuren 1990b, 1990c). It is intended now to explore issues such as lack of realizability and compatibility with the two-component limit as the wall is approached. These second-moment closure models will also be extended for application to compressible flows through the incorporation of models for additional terms which arise such as pressure-dilatation, density-velocity, density-temperature correlations, etc. The models are to be incorporated in existing LU and ADI codes for both 2D and 3D flows.

REFERENCES

Demuren, A.O.: Calculation of Turbulence-Driven Secondary Motion in Ducts With Arbitrary Cross-Section. AIAA J., vol. 29, 1991 (in press). (Also, NASA TM-102142).
Demuren, A.O.: Calculation of Turbulent Flow in Complex Geometries With a Second-Moment Closure Model. ASME Fluids Engineering Conf., Toronto, 1990.
Demuren, A.O.: Calculation of 3D Turbulent Jets in Crossflow With a Multigrid Method and a Second-Moment Closure Model. Engineering Turbulence Modelling and Experiments, W. Rodi and E.N. Ganic, eds., Elsevier, 1990, pp.   (Also, NASA TM-103159).

Demuren, A.O.: Characteristics of 3D Turbulent Jets in Crossflow. 12th Symp. on Turbulence, Rolla, MO, 1990.
* Now at Old Dominion University

Peter R. Eiseman, Columbia University

The consulting activities at ICOMP comprised developmental work on guiding the sequence of TURBO codes, the first part of a lecture sequence on adaptive grid generation, and the initiation of research for generating pointwise distributions on curves. The TURBO codes represent a continuing effort at NASA Lewis under the direction of Dr. Yung K. Choo. Each code is based upon the control point form (CPF) of algebraic grid generation that was developed by the consultant. Moreover, several technical papers co-authored with Yung K. Choo and others have resulted.

While a substantial effort has been directed toward the interactive applications of the control point form of algebraic grid generations, as in TURBO, a new effort is being directed toward the more automatic application. A prominent part of this is the task of creating a fully adaptive strategy whereby the control points automatically adjust their positions to provide a dynamic resolution of an evolving numerical solution to a fluid flow problem. In conjunction with the full automation of the solution adaptive procedures, there are also a number of partially automated situations whereby some subcollection of control points are to be moved. Such partial cases are considered for a broader variety of purposes. Not only is the provision of appropriate clustering important, but also the improvement in the grid structure, in a metric sense, is important.

The first of a sequence of lectures on adaptive grid generation was successfully given and covered many of the basic elements in the somewhat uncluttered mathematical framework of one dimension. The next lectures will fill out the one-dimensional story, extend those results into multiple dimensions in successive stages, and deal with the fundamental techniques for coupling the grid to a PDE solver.

J. S. B. Gajjar, University of Exeter

I looked at the problem of a single oblique mode propagating in a 2D compressible boundary layer. By taking suitable scalings the downstream evolution of the mode can be shown to governed by the unsteady nonlinear critical layer equations similar to those arising in many shear layer flows. This is true for heated/cooled boundary layers in both subsonic and supersonic flow provided that the long wavelength approximation (based on the inviscid scalings) is used. The work applies equally to flat plate boundary layers with adiabatic wall conditions if the Newtonian approximation if made, but only for low Mach numbers. I have developed a code for solving the unsteady nonlinear critical equations using a method based on Fourier series decomposition and Chebychev collocation. This method is very different from that used by Dr. Goldstein's group, and most of my time was spent in trying to reproduce some results for some test cases. The code, which was initially written to run on a AMT DAP-510 parallel processor, was successfully modified to run on the Cray-YMP and most of my results were found to be in excellent agreement with those obtained by the people here. However, there appears to be one or two discrepancies in one test case and these have, so far, not been resolved. The code will eventually be used to perform several parametric case studies and the work will be written up and submitted for publication at some stage.

Thomas Hagstrom, SUNY, Stony Brook*

While at ICOMP this June, I carried out research on various problems in the numerical analysis of unsteady fluid flows with special emphasis on the issue of accurate numerical radiation conditions. Among the accomplishments of the three week stay were:

Completion of a survey and comparative study of boundary conditions for the wave equation in an exterior, two-dimensional geometry. Theoretical considerations predict difficulties in attaining great accuracy for long time computations and our numerical experiments confirm this prediction.

With S.I. Hariharan we continued our work on boundary conditions for the Euler equations. Our concentration was on the translation and implementation of higher order conditions which have been proposed for the wave equation to the derivation of high order conditions on the pressure.

Discussed with J. Goodrich of NASA Lewis and A. Demuren of ICOMP the implementation of previously developed boundary conditions for incompressible flows. We plan to submit these to a mini-symposium on the testing of outflow boundary conditions to be held next June.

I also discussed and planned projects in other areas of computational fluid mechanics. With J. Goodrich, a long range plan for the development and application of an incompressible, low Mach number algorithm for the simulation of complex, unsteady flows with chemical reactions was outlined. The problem of studying numerical stability and transition in pipe flows was discussed with D.Ashpis of NASA Lewis.
* Now at the University of New Mexico

Charles Hall and Thomas Porsching, University of Pittsburgh

The computer code ALGAE has been developed at the University of Pittsburgh by the investigators over the past 14 years. This program treats incompressible, thermally expandable or locally compressible flows in complicated flow regions including multiply connected regions in two dimensions. During this last year, the investigators modified the code to allow for heat conduction between the fluid and an embedded solid, added time dependent boundary conditions and modified the constitutive relations. These modifications provided an analysis tool to better simulate the thermodynamic behavior of the working gas in Stirling cycle engine components. Several simulations of the heater-regenerator-cooler subsystem were made using ALGAE on the NASA Ames Cray Y/MP. Video animations were made from several of these runs. The results were analyzed and a report was prepared and presented at the IECEC meeting in Reno.

REFERENCE

Hall, C.A., et al.: Multidimensional Computer Simulation of Stirling Cycle Engines. 25th Intersociety Energy Conversion Engineering Conference, Vol. 5, AIChE, 1990, pp. 407-411.

S. I. Hariharan, University of Akron

Computational Methods for Compressible Gas Dynamics Problems (with T. H. Hagstrom, University of New Mexico)

We started our analysis by considering anisotropic wave equations in both two and three dimensions. The goal was to derive uniform far-field expansions of the solution which are generalizations of the Friedlander expansion valid for isotropic media. Our derivation is based on the use of "wave front" coordinates related to the geometrical acoustics approximation. An infinite series has been obtained recursively from the radiation field. The derivations for conditions for the convergence of the series is in progress. Using this representation we have derived boundary conditions for problems in unbounded regions governed by the Euler equations. Also, we have tested the effectiveness numerically. We are in the process of completing the theory to investigate the well-posedness of the problem governed by the Euler equations. Also, extension of our theory to viscous flows is under investigation.

Unsteady Aerodynamics (with J. R. Scott, NASA Lewis Research Center)

Calculation of unsteady vortical disturbances around the airfoils is the main goal. The problem is governed by a time dependent convective wave equation in an open domain. The open domain problem, when truncated into a finite domain by artificial boundaries, requires approximate boundary conditions that simulate the behavior at infinity. Approximate boundary conditions are derived based on our asymptotic theory (described above). A numerical solution procedure for the time domain calculation has been derived. Preliminary comparisons indicate good agreements with solutions of a corresponding frequency domain problem. These results are to appear in **The Journal of Computational Physics**. In this paper, one of the disturbing aspects was the accuracy of results at low frequencies. Treatment to address this issue is in progress.

Absorbing Boundary Conditions for the Wave Equation; Low Frequency Corrections (with T. Hagstrom, University of New Mexico and R. C. MacCamy, Carnegie-Mellon University)

The long time behavior of the solution is a question that often arises in time-dependent problems and their numerical simulations. The study of these poses a strong challenge in constructing numerical radiation conditions, particularly in the light of the low-frequency (an example is the gust problem described above) long time nonuniformities inherent in the asymptotic analysis. In a recent paper, Engquist and Halpern study the effect of adding a low-frequency correction to a standard absorbing boundary condition. We have derived a more general class of boundary conditions which encompasses both the low- and high-frequency behavior of the exact boundary operator for the wave equation in two dimensions. We are examining the well-posedness of the resulting initial-boundary value problem, the efficient implementation of the nonlocal conditions, and the long time behavior of the error. (A preliminary version of our results was presented at the SIAM National meeting in Chicago, July 1990.)

J. Mark Janus, Mississippi State University

During my two week visit at ICOMP/NASA Lewis, I investigated the accuracy of relative-motion domain decomposition interfaces. My test-bed was a one-dimensional shock problem modeled to allow solution variation along the shock front, i.e., modeled in two dimensions. The lower half of the tube was discretized with a stationary grid, while the upper half was discretized with a uniform dynamic grid moving at the shock speed. Thus, the configuration has a shock which spans the subdomain interface. Across this interface the solutions which evolve in each subdomain must communicate. In the upper domain, since the grid is moving with the shock, the shock is "seen" by the flow solver as a stationary wave (discontinuity). In the lower subdomain the shock is "seen" as an unsteady wave; thus, one must rely on the numerical algorithm to propagate this wave accurately.

As a preliminary finding, it was observed that any resolution difference between the way a numerical algorithm represents steady phenomena as compared to unsteady phenomena will give rise to nonphysical flow behavior at an interface such as the one described here. Actually, if the unsteady accuracy depends on the temporal stepsize, wavespeed, and/or wave strength (which it does), nonphysical behavior can be expected at relative-motion interfaces. Additional studies need to be performed to better understand the "numerical physics" surrounding these inter-researchers utilizing relative-motion domain decomposition.

During my stay, I constructed the software necessary to perform several studies into the physics of relative motion interfaces. My initial suspicions have been borne out in my early test runs, but much, much more indepth study is in order. I plan to continue this work until I have reached satisfactory conclusions as to the nonphysical effects of relative-motion interfaces, what causes the effects, and how to best eliminate or diminish them.

Bo-Nan Jiang, (Postdoctoral) University of Texas-Austin

In this research we first introduce the least-squares (L_2) finite element method for 2D steady-state pure convection problems with smooth solutions. We prove that the L_2 method has the same stability estimate as the original equation, that is, the L_2 method has better control of the streamline derivative. Numerical convergence rates are given to show that the L_2 method is almost optimal. Then we use this L_2 method as a framework to develop an iteratively reweighted L_2 finite element method to obtain a least absolute residual (L_1) solution for problems with discontinuities. This L_1 finite element method captures 2D discontinuity in bands of elements that are only one element wide on both coarse and fine meshes. The solution of this method has no smearing and no oscillation, and has superior accuracy. We also devised a robust reweighting strategy to obtain the L_1 solution in few iterations. A number of examples solved by using triangle and bilinear elements are presented.

S.-W. Kim (Postdoctoral) University of Texas - Arlington

Numerical calculation of a circular jet exhausting into a uniform cross flow is in progress. The numerical method is based on finite volume method with pressure-staggered mesh. The turbulence is described by a multiple-time-scale turbulence model.

Numerical results show that the cross-jet flow is characterized by a strong inequilibrium turbulence field. In front of the cross-jet, the ratio of the production rate to the dissipation rate of turbulent kinetic energy varies rapidly in the region, while, in the wake region of the cross-jet, the turbulence is in a more equilibrium state even though the turbulence level in the wake region is by far higher than that of the front region. The calculated mean velocity profiles and the turbulent kinetic energy profiles are in good agreement with the measured data. Such an accurate prediction of the flow field is attributed to the capability of the multiple-time-scale turbulence model to resolve the effect of a strong inequilibrium turbulence field on the mean flow field. As yet, it has not been shown that single-time-scale turbulence models (such as k-ϵ or Reynolds stress turbulence models) can resolve inequilibrium turbulence fields.

The numerical results also show that the jet and the cross-flow interact very strongly with each other at the jet exit. The strong interaction between the jet and the cross-flow suggests that the computational flow domain needs to be located in the upstream region of the circular jet to test the predictive capability of any turbulence model or to obtain accurate numerical results. The calculated velocity profiles in the vicinity of the jet exit are in good agreement with the measured data.

B. P. Leonard, University of Akron

Work continued on the development of higher-order convection methods for flows containing discontinuities. Of particular interest is the question of computational efficiency (best accuracy for a given cost): which is more efficient, an inexpensive low-order computation on a very fine grid or a more expensive (per grid-point) high-order method on a relatively coarse grid? Is there an optimum order in this respect? Numerical experiments of scalar convection (containing discontinuities) in variable velocity fields (e.g., the Smith-Hutton problem) have shown that the most efficient scheme (used globally) is either third- or fifth-order upwinding. First- and second-order methods (or hybrid combinations) are much less efficient because of the excessively fine grid required for a given accuracy; alternatively, the expense of using higher (than fifth) order is not offset by a concomitant coarsening of the grid. But, by far the most efficient strategy is to use adaptive stencil expansion: using third-order upwinding in "smooth" regions (the bulk of the flow domain) and automatically switching to fifth- or seventh-order methods in local regions requiring higher resolution (by their nature, such regions involve only a relatively small number of grid points). This work, co-authored with Simin Mokhtari, was reported in ICOMP-90-12 (NASA TM-102568) and published under the title "Beyond First-Order Upwinding: The ULTRA-SHARP Alternative for Non-Oscillatory Steady-State Simulation of Convection" in the International Journal for Numerical Methods in Engineering, Volume 30, pages 729-760 (1990).

Avi Lin, Temple University

Parallel computing as a tool to resolve delicate flow phenomena is a very important issue in the area of advanced CFD. Specifically, we try to solve numerically the flow equations on an MIMD parallel distributed computational environment. The concept of the "Master-Slave" is used to generate a parallel numerical scheme for the Navier-Stokes equations, which are also computationally efficient on the parallel machines. This approach results in two basic possibilities for the coarsest parallel-grain of the numerical scheme: the parallel domain decomposition and the parallel operator decomposition. The former is quite a straight forward approach which is, basically, an explicit scheme over the processors. Although it is easily applied, it usually results in a very large iterative matrix that most of the time is sparse, and thus some additional (serial) technique has to be employed in order to make a useful implementation on the parallel machine.

We have concentrated on the parallel operator decomposition which allows one to formulate implicit parallel numerical schemes and yet keeps the communication overhead to a minimum. We have explored this approach on different levels of the computations: on the basic matrix inversion level, on the higher level of block-tri-diagonal linear systems as well as block-banded linear systems. The implementation for parallel solving boundary value problems and elliptic problems is as far as we got. One of the interesting implementations that was found was for the parallel parabolized Navier-Stokes equations.

The parallel operator decomposition enforces changes in the standard numerical schemes. For example, it is recommended to minimize the bandwidth of the iterative matrix (or the computational stencil) for high parallel computational efficiency. Using a new high order up-wind scheme which is spread over the standard three grid points enables one to get an accurate solution and still maintain an efficiency of about 75 percent. We proved theoretically that this is more or less as much efficiency as one can get when solving elliptic problems on a Hypercube architecture.

This research is a joint activity between ICOMP and the parallel computing team of IFMD. The implementations were done on the Hypercluster machine which was proven to be well suited for the CFD problems.

The turbulent Reynolds Stress dynamic equations and the Tensorial Length scale dynamic equations are not being implemented on the system. The operator decomposition technique allows for a tight coupling between the turbulent quantities (i.e., the Reynolds stress and the appropriate length scale) while the decomposition is done along some geometrical directions (e.g., the system coordinates or the eigen-directions of the flow).

W. W. Liou, (Postdoctoral) Pennsylvania State University

The goal of my research is to develop Reynolds stress closures for compressible flows. The classic second-order Reynolds stress closures proposed by Launder, et. al, were developed for incompressible flows. In attempts to calculate compressible flows, extensions of these incompressible models are often made rather than starting with the exact Reynolds stress equations. However, as the Mach number of a turbulent flow increases, the effects of fluctuating thermodynamic state variables, such as density and pressure, on the turbulence structure and dynamics grow as well. Therefore, simple modifications of incompressible formalisms can hardly provide a rational approach for modelling complex compressible turbulence, not to mention applying them to engineering flow calculations. The approach we will take here is to re-examine the exact compressible turbulence equations. This will be aided by order of magnitude analyses and the results of Direct Numerical Simulations and experiments. Then, the next step is to propose rational models to describe the important mechanisms of compressible turbulence. We hope that these models will not only represent turbulence physics, but also provide feasible tools for modeling turbulence in engineering calculations.

J. Mathew, (Postdoctoral) Massachusetts Institute of Technology

Recently, Goldstein, et. al (ref. 1) studied the development of small steady perturbations to a uniform flow along a flat plate. Corresponding to spanwise variations of these perturbations, there appears a spanwise steepening of velocity profiles in the boundary layer, rapid growth in boundary-layer thickness and the wall shear stress vanishes, suggesting early transition. I have completed the formulations and program development for some of the associated heat transfer problems, including compressible flows. Data from these calculations can be tested against experiments that may be done here at Lewis; VanFossen and Simoneau (ref. 2), of NASA Lewis, have already conducted experiments involving a similar incident flow past a circular cylinder.

I have also begun examining the action of trailing vortex sheets behind an array of thin airfoils on the flat plate boundary layers.

REFERENCES

Goldstein, M.E.; Leib, S.J.; and Cowley, S.J.: The Effect of Small Streamwise Velocity Distortion on the Boundary Layer Flow Over a Thin Flat Plate With Application to Boundary Layer Stability Theory. J. Fluid Mech. sub judice. (Also, NASA TM-103668)
VanFossen, G.J., Jr. and Simoneau, R.J.: Preliminary Results of a Study of the Relationship Between Free Stream Turbulence and Stagnation Region Heat Transfer. ASME Paper 85-GT-84, Mar. 1985.

Mohammd A. Mawid, (Postdoctoral) University of Illinois-Chicago

Full numerical simulations of a gaseous laminar diffusion flame and a diffusion flame supported by liquid fuel droplet streams have been performed using the KIVA-II computer code. The computations were two-

dimensional axisymmetric. The inflow velocities of both the gas and the droplet streams were specified to be slightly less than the experimentally measured laminar speed such that a flame can be stabilized. Further, chemical kinetics were modelled by a one-step reaction scheme and no equilibrium reactions (i.e., fast reactions) were considered. The effects of monodisperse liquid fuel streams, polydisperse fuel streams (i.e., variation of SMD), number of liquid fuel streams, and inlet gas and droplet velocities on the flame structure and shape were studied.

In conjunction with the diffusion flame analyses, computations of turbulent combusting sprays in elliptic flows, which are being currently investigated experimentally under the supervision of Dr. D. Bulzan at NASA Lewis Research Center, has been initiated. A two-equation turbulent flow model with a one-step chemical reaction model is employed. The initial profiles of the mean gas-phase velocities, mixture mass fractions, temperature, turbulent kinetic energy and its dissipation rate, drop size distribution, and drop velocity distribution with swirl are chosen such that a flame can be stabilized. The computations of the turbulent combusting spray for various measured profiles and the effects of important parameters will be continued next year during my association with ICOMP.

A. F. Messiter, University of Michigan

Two studies of shear layer instabilities have been initiated during my four week visit to ICOMP. In each case, the description of a nonlinear, critical layer is closely related to previous work of M. Goldstein, L. Hultgren, and S. Leib. The first study is concerned with an incompressible shear layer subjected to a slowly growing disturbance having periodic spanwise variation on a scale large in comparison with the disturbance wavelength. The 2D formulation is modified by the addition of a vortex-stretching term in the vorticity equation, as well as a spanwise momentum equation. The analytical formulation is nearly completed and should be followed by numerical calculations. A study of supersonic shear layers has also been started, with the goal of determining how (and whether) nonlinearity in the critical layer is related to nonlinearity in the external flow. It appears that a limiting case exists in which the two nonlinearities appear simultaneously.

Vittorio Michelassi, (Postdoctoral), University of Florence, Italy

During my three months stay at ICOMP, I worked on lower order turbulence models. While higher order turbulence closures involve more physics in the modelling of turbulence, at the present time, lower order closures still represent a good compromise between accuracy and computational efficiency; and, they often produce results in reasonable agreement with experiments. Among the various formulations available in the literature, I have chosen for a full comparison those that allow integration to the wall avoiding the so called "wall-functions". These forms, known as "Low Reynolds number form" (LR) of two equation models account for viscous effects in the wall region, thereby, providing better agreement with experiments for wall driven flows.

Unfortunately these models are both numerically stiff and require information about the flow geometry using the dimensional or nondimensional wall distance. This need decreases the model generality and often prevents the use of the models in presence of stagnation regions or complex geometries.

In view of this, a new formulation that proved not to be stiff and that does not need the wall distance is proposed starting from the two-equation model presented by Shih and using direct numerical simulation results. Predictions obtained with this formulation are compared with those of seven other LR forms in a fully developed channel flow configuration. The new formulation gives very good accuracy even though it does not need any information about the flow geometry. The eight models are then applied to the computation of the flow past a hill in the presence of a small recirculation bubble. The implicit solver used for all the models was explicitly designed for turbulence model testing.

The algorithm is based on a Beam-Warming type solver with the artificial compressibility method, and a centered finite difference discretization scheme. Mesh refinement was carried out until artificial damping was not needed any more. Again, the computed results prove that with the new formulation it is possible to obtain the same accuracy and fit with experiments provided by models that need the dimensional or nondimensional wall distance.

Subodh K. Mital, (Postdoctoral) Case Western Reserve University

Microfracture (i.e., fiber/matrix fracture and fiber-matrix interface debonding) in composites has been computationally simulated. A computational simulation procedure has been developed, as part of this activity. The procedure uses 3D finite element analysis and global strain energy release rates to predict the microfracture process and identify/quantify the hierarchy of respective fracture modes in a composite. This procedure has been applied to predict the microfracture process in a unidirectional metal matrix composite subjected to various types of mechanical loading. Another novel procedure to computationally simulate the fiber pushout process has been developed. In this simulation, the interphase material is replaced by an anisotropic material with greatly reduced shear modulus in order to simulate the fiber pushthrough process using a linear analysis. Such a procedure is easily implemented and is computationally very effective.

Work in progress includes simulation of microfracture in cross-plied and angle-plied laminates, simulation of microfracture in hybrid composites, thermally-driven microfracture, and finally to develop a unified approach to evaluating composite microfracture. Also, a flatwise test method is being developed to measure, in-service, the loss of strength and stiffness in composites due to the microcracking that develops because of thermal/mechanical cyclic loading.

REFERENCES

Mital, S.K.; and Chamis, C.C.: Fiber Pushout Test: A Three-Dimensional Finite Element Computational Simulation. NASA TM-102565, 1990.
Mital, S.K.; Caruso, J.J.; and Chamis, C.C.: Metal Matrix Composites Microfracture: Computational Simulation. NASA TM-103153, 1990.

Christophe Pierre, University of Michigan

The basic purpose of this research is the development of computational methods for understanding and predicting the effects of unavoidable blade-to-blade dissimilarities, or mistuning, on the dynamics of nearly cyclic bladed-disk assemblies. This topic is of importance as mistuning has been shown to increase the forced response amplitudes of some blades drastically (by several hundred percent!) and even to lead to blade failure.

Our research has focused primarily on the topics described below:

We have developed a true measure of sensitivity to mistuning for generic structural models of turbomachinery rotors. Because mistuning is known in a statistical fashion only, our measure of sensitivity had to be probabilistic. The primary result of our research is that we showed the mean of the second-order eigenvalue perturbation to be a very good descriptor of sensitivity. Specifically, we proved that first-order perturbation terms completely overlook high sensitivity to mistuning and that second-order terms must be included. Furthermore, we obtained analytical expressions for the sensitivity of simple models of blade assemblies. This makes our proposed sensitivity descriptor a very cost-effective tool for identifying parameter ranges where small mistuning has large effects. Indeed, we found the cost of our measure of sensitivity to be negligible compared to that of Monte Carlo simulations. These research results will be described in an ICOMP publication whose writing is nearing completion (1).

We have examined the effects of mistuning on the aeroelastic modal characteristics of blade assemblies. We have shown that the aeroelastic response of blade assemblies is highly sensitive to small mistuning. In particular, the aeroelastic modes of the system loose their constant interblade phase angle characteristic when mistuning is introduced: they become severely localized to a few blades and no pattern can be discerned for the interblade phase angle. Furthermore, the root locus of the aeroelastic eigenvalues loses the regular pattern that characterizes the tuned system to become apparently randomly scattered for small mistuning. We have developed perturbation schemes that explain and predict this sensitivity to mistuning. The original contributions of our study lie in the evidence of these new phenomena and in the generality of the mistuning trends and phenomena uncovered: our results are qualitatively valid for most typical blade assemblies. An ICOMP paper that describes these results is in preparation (2).

We have applied these general findings to the complete model of an actual mistuned high-energy turbine. Our results show an <u>extreme</u> sensitivity to mistuning for the aeroelastic modes of the turbine. The transition from constant-interblade-phase-angle modes to localized modes is very rapid and exhibits a complex behavior of the eigensolution. Severe localization occurs for blade frequency mistuning of approximately 0.1 percent — a mistuning level clearly unavoidable for a real turbine. Results are being written up as an ICOMP paper (3).

Future research will focus on extending these studies to define a stochastic measure of sensitivity for aeroelastic systems and to examine the forced response problem.

REFERENCES

Pierre, C.: Probabilistic Measure of Root Sensitivity to Uncertainties in Structural Dynamics. To be submitted to the ASME Journal of Vibration and Acoustics.

Pierre, C.; and Murthy, D.V.: Aeroelastic Modal Characteristics of Mistuned Blade Assemblies: Mode Localization and Loss of Eigenstructure. AIAA Paper 91-1218, Apr. 1991.

Pierre, C.; Smith, T.E.; and Murthy, D.V.: Localization of Aeroelastic Modes in Mistuned High-Energy Turbines. Submitted to the AIAA 27th Joint Propulsion Conference, Sacramento, June 1991.

Richard H. Pletcher, Iowa State University

Work was initiated with Philip Jorgenson on the simulation of internal viscous flows using unstructured grids. The accurate and economical simulation of flows in complex geometries is one of the remaining major challenges in computational fluid dynamics. When conventional structured grids are used, the effort associated with establishing a satisfactory grid for a flow field that includes complex geometry is often a large fraction of the total effort required to simulate the flow. The fraction of the effort devoted to grid generation is even greater for unsteady problems in which relative motion of objects within the flow field occurs. An alternative procedure is to use computational cells in an unstructured mesh that can be adapted to conform to a complex geometry and readily subdivided to resolve regions of steep gradients in the flow and thermal variables. The unstructured strategy shows much promise, but much work remains to be done in order to make the approach economically competitive for viscous flows, especially in three dimensions. The objective of the present study is to develop an accurate and economical simulation strategy for the solution of the Navier-Stokes equations utilizing unstructured grids that is particularly well suited for turbomachinery applications. A procedure to generate unstructured meshes in two dimensions has been developed, and work is underway to develop an efficient numerical scheme for solving the conservation equations.

Numerous discussions were held with Lewis personnel including Meng-Sing Liou, Russ Claus, Robert Simoneau, Steve Hippensteele, and John Goodrich on research topics of mutual interest, particularly flow solvers for low-speed, compressible flows and combustion applications, unsteady flows, and turbulence modeling.

Thomas Porsching, University of Pittsburgh

See HALL, Charles above.

T.-H. Shih, Stanford University (CTR)

The objectives of this work are: (1) examine the performance of existing two-equation eddy viscosity models and develop better models for the near-wall turbulence using direct numerical simulations of plane channel and boundary layer flows, (2) use the asymptotic behavior of turbulence near a wall to examine the problems of current second-order closure models and develop new models with the correct near-wall behavior of these models, and (3) use Rapid Distortion Theory (RDT) to analytically study the effects of mean deformation (especially due to pure rotation) on turbulence, and to obtain analytical solutions for the spectrum tensor, Reynolds stress tensor, anisotropy tensor and its invariants, which can be used in the turbulence model development.

1. **k-ϵ model**: The k-ϵ model is still the most widely used model for computing engineering flows. We have examined the near-wall behavior of various eddy viscosity models proposed by different researchers,

and studied the near-wall behavior of terms in the k-equation budget. We found that the modelled eddy viscosity in many existing k-ϵ models does not have correct near-wall behavior and the pressure transport term in the k-equation is not appropriately modelled. Based on the near-wall asymptotic behavior of the eddy viscosity and the pressure transport term in the k-equations, a new set of improved closure models has been obtained. In addition, a modelled equation for the dissipation rate is derived more rationally.

2. **Second order modelling of near-wall turbulence**: The main emphasis is on developing a near-wall turbulence model for the velocity pressure gradient correlation and the dissipation tensor in the Reynolds-stress equation. A modelled dissipation rate equation is also derived more rationally. Near the wall, a reduction in velocity fluctuations normal to the wall become significant. Because of this wall effect, the viscous diffusion term in the Reynolds-stress equations becomes the leading term and it must be properly balanced by the other terms. We have used this as a model constraint for developing a model for the pressure and dissipation terms. To test the models, a fully developed channel flow and boundary layer flows are chosen as the test flows, for which direct numerical simulations and experimental data are available for comparison. The modelled Reynolds stress equations for the channel flow are one-dimensional and steady, and for boundary layer flows are two-dimensional steady. Therefore, model testing will be very accurate.

3. **Second order modeling of a three-dimensional boundary layer**: A study of three-dimensional effects on a turbulent boundary layer were achieved by direct numerical simulation of a fully developed turbulent channel flow subjected to transverse pressure gradient (the work was done at CTR, Stanford). The time evolution of the flow was studied. The results show that, in agreement with experimental data, the Reynolds stresses are reduced with increasing three-dimensionality and that, near the wall, a lag develops between the stress and the strain rate. In addition, we found that the turbulent kinetic energy also decreased. To model these three-dimensional effects on the turbulence, we have tried different second order closure models. None of the current second order closure models can predict the reductions in the shear stress and turbulent kinetic energy observed using direct numerical simulations. Detailed studies of the Note: first equation on this page is missing a sq. rt. symbol before the R_t!! Reynolds-stresses budgets were carried out. One of the preliminary conclusions from these budget studies is that the velocity pressure-gradient term in the normal stress equation (v^2) plays a dominant role in the reduction of shear stress and kinetic energy. These budgets have been used to guide the development of better models for three-dimensional turbulent boundary layer flows.

4. **The effect of rotation on turbulence**: In addition to the above studies of second order closure models, we have carried out some RDT analysis on simple homogeneous turbulent flows. An order of magnitude analysis shows that under the condition of $S(q^2)/\epsilon \gg R_t$, the equations for turbulent velocity fluctuations can be approximated by a linear set of equations, and if $S(q^2)/\epsilon \gg R_t^{3/4}$, then the turbulent velocity equations can be further approximated by an inviscid linear equation. Therefore, RDT can be used to analytically study some very basic turbulent flows such as homogeneous shear flows, irrotational strain flows and pure rotational flows. This work focuses on the effect of rapid rotation on turbulence using RDT. We have obtained analytical expressions for velocity, the spectrum tensor, Reynolds-stress, the anisotropy tensor and its invariants. The solutions show that the turbulence is strongly affected by the rapid rotation. Using RDT, we can calculate the rapid pressure-strain term exactly and we can obtain very useful information for developing corresponding turbulence models.

Avram Sidi, Technion, Israel Institute of Technology

The research on vector valued rational approximations and their applications that began during the Summer of 1989 was continued and expanded to cover meromorphic functions with multiple poles (as opposed to simple poles). The results of this research are being written up and will be published in the near future. Among the results obtained, there is a theorem on uniform convergence of the approximations to the function being approximated in the complex plane. Two other results show convergence of poles and corresponding residues of the rational approximations to those of the function being approximated. The application of these results to the algebraic eigenvalue — eigenvector problem is immediate as the eigenvalues and eigenvectors correspond to the poles and residues of some meromorphic function defined in terms of power iterations. The application to time-periodic, steady-state problems can also be handled if one observes that the asymptotic structure of the time-marching finite difference approximations is very similar to that present in the algebraic eigenvalue problems.

The methods proposed have been tested successfully by Mark L. Celestina on the algebraic eigenvalue problem.

Research began on the application of vector extrapolation methods with frozen weights. This was tried on linear and nonlinear problems with success. This approach is advantageous when the iterative procedure that produces the vector sequence is very inexpensive.

A computer program for vector acceleration written at Technion, Haifa, was given to different users and collaboration was set up with two. The program was used in conjunction with a combustion code by Dr. Shaye Yungster and a fully developed incompressible channel flow code by Dr. Hsiao Kao and very good acceleration was obtained in both cases.

Joseph L. Steger, University of California-Davis

A major criticism leveled against the chimera approach for flow simulation is that nonconservative interpolations are often used in practical three-dimensional computations to update interface boundaries. The fact that interpolation is generally used to connect grids implies that conservation is not strictly enforced. For many practical applications it is difficult to devise a situation where this error is overall-significant since conservation is strictly maintained at all points in the domain (assuming the solver is a conservative one) except at a small number of interface boundaries. Nevertheless, a solution to this problem is needed to allay the fear that it will cause the chimera method to break down on some future critical problem.

During my stay at ICOMP I studied several approaches for treating the problem of nonconservative interfaces. I also worked with Meng Liou to understand the approach that he and Moon devised for treating this problem.

1. Flux Balancing Schemes.
   Special schemes that maintain conservation at interface boundaries for overset grids have been devised by M. Berger and by Y. Moon and M. Liou. These schemes, however, have not been implemented into three-dimensional codes because of their complexity. Liou has now significantly simplified the approach that he and Moon have taken, and this appears to be the most accurate procedure under current development.

2. Delta Form Interpolations.

   In lieu of interpolating the function $\vec{Q} = (\rho, \rho u, \ldots)^t$ to the interface boundary points, one can interpolate

   delta-time quantities, specifically $\vec{Q}^{n+1} - \vec{Q}^n$. This simple modification to interpolation was suggested by J. Benek and it has been used in numerous Calspan chimera resutls for the past several years. Preliminary analysis indicates that interpolating the delta quantity on interface boundaries ensures space-time conservation over the global field.

To illustrate this, consider the unsteady equation

$$\partial_t u + \partial_x (au) = 0 \quad \text{where} \quad a = a(x, t, u) \tag{1}$$

A simple difference scheme for equation (1) is given by

$$u_j^{n+1} - u_j^n + \alpha_{j+1}^n u_{j+1}^n - \alpha_{j-1}^n u_{j-1}^n = 0 \tag{2}$$

where

$$\alpha = \left(\frac{a \Delta t}{2 \Delta x} \right)$$

The differencing given by equation (2) is unstable. A Lax-Wendroff scheme which uses the same data stencil could be used, but as only the **conservative** properties of a scheme are of interest here, the simpler unstable scheme suffices.

Let equation (2) be differenced on the uniform mesh

$$\begin{array}{ccccccccc} \circ & \bullet & \bullet & \bullet & \bullet & \bullet & \bullet & \bullet & \circ \\ 1 & 2 & 3 & 4 & 5 & 6 & 7 & 8 & 9 \end{array}$$

with the numbers 1 to 9 indicating index location.

Summing the difference equations, equations (2), over all space and time (column summation) leads to

$$\sum_{n=0}^{N} \left[-\left(\alpha_1^n u_1^n + \alpha_2^n u_2^n\right) + \left(\alpha_8^n u_8^n + \alpha_9^n u_9^n\right)\right] = \sum_{j=2}^{8} \left(u_j^{N+1} - u_j^0\right) \tag{3}$$

The left hand side represents space flux in an out of the (entire space time) domain while the right hand side represents the time flux in and out of the domain. This is a conservative differencing scheme and all interior flux terms cancel out in the summation process.

Consider now, the two overset meshes:

$$\begin{array}{ccccc} \circ & \bullet & \bullet & \bullet & & \bullet \\ 1 & 2 & 3 & 4 & & 5 \end{array}$$
$$\begin{array}{cccc} \bullet & \bullet & \bullet & \circ \\ 6 & 7 & 8 & 9 \end{array}$$

Because of the decomposition into two meshes, interface boundary conditions are given at the interior end points 5 and 6. Let the solution at the interface points 5 and 6 be obtained via simple interpolation of the time **delta** quantities. That is, the solution at points 5 and 6 is obtained from

$$\left(u_5^{n+1} - u_5^n\right) - \beta\left(u_6^{n+1} - u_6^n\right) - (1 - \beta)\left(u_7^{n+1} - u_7^n\right) = 0 \tag{4a}$$

$$\left(u_6^{n+1} - u_6^n\right) - \bar{\beta}\left(u_4^{n+1} - u_4^n\right) - \left(1 - \bar{\beta}\right)\left(u_5^{n+1} - u_5^n\right) = 0 \tag{4b}$$

where $0 \le \beta \le 1$ and $0 \le \bar{\beta} \le 1$. Summing the difference equations (governing equations and interpolations) over all space and time (column summation) leads to

$$\sum_{n=0}^{N} \left[-\left(\alpha_1^n u_1^n + \alpha_2^n u_2^n\right) + \left(\alpha_4^n u_4^n + \alpha_5^n u_5^n\right) - \left(\alpha_6^n u_6^n + \alpha_7^n u_7^n\right) + \left(\alpha_8^n u_8^n + \alpha_9^n u_9^n\right)\right]$$
$$= -E_I\left(u^{N+1} - u^0\right) + \sum_{j=2}^{4} \left(u_j^{N+1} - u_j^0\right) + \sum_{j=7}^{8} \left(u_j^{N+1} - u_j^0\right) \tag{5}$$

where

$$E_I\left(u^{N+1} - u^0\right) = \left(u_5^{N+1} - u_5^0\right) - \beta\left(u_6^{N+1} - u_6^0\right) - (1 - \beta)\left(u_7^{N+1} - u_7^0\right)$$
$$+ \left(u_6^{N+1} - u_6^0\right) - \bar{\beta}\left(u_4^n - u_4^0\right) - \left(1 - \bar{\beta}\right)\left(u_5^{N+1} - u_5^0\right) \tag{6}$$

represents the interpolation error at only the initial and final time levels.

Consider now that on the mesh points 1 to 5 the difference equations, equation 2, are conservative and summation over space and time leads to

$$\sum_{n=0}^{N} \left[-\left(\alpha_1^n u_1^n + \alpha_2^n u_2^n\right) + \left(\alpha_4^n u_4^n + \alpha_5^n u_5^n\right)\right] = \sum_{j=2}^{4} \left(u_j^{N+1} - u_j^0\right) \tag{7}$$

Likewise on the mesh points 6 to 9 the difference equations are conservative and summation over space and time leads to

$$\sum_{n=0}^{N} \left[-\left(\alpha_6^n u_6^n + \alpha_7^n u_7^n\right) + \left(\alpha_8^n u_8^n + \alpha_9^n u_9^n\right) \right] = \sum_{j=7}^{8} \left(u_j^{N+1} - u_j^0\right) \tag{8}$$

If equations 7 and 8 are substracted from equation 5 one obtains

$$0 = -E_I\left(u^{N+1} - u^0\right)$$

Thus interior to the initial and final time levels, the differencing is space-time consevative. The error in conservation due to interpolating the time delta quantitites will only show up at the initial and final states — regardless of how many time steps are taken. This is acceptable and not unlike interpolating a flux to a grid cell face. By interpolating time-delta quantities erors in conservation cannot accumulate.

3. **Shock Fitting of Interface Boundaries.**

Fitting of shocks crossing through chimera interface boundaries also appears feasible. This is because the interface boundary is a well defined surface with known flow conditions to either side. Thus if a shock wave can be identified to either side of the interface boundary, values on the interface boundary can be updated using shock jump conditions. This appears quite feasible since only a small number of surfaces need be tested for shock crossing, and the testing is based on observing an adjacent shock in the interior grid. While shock identification is not easy in general, some software is available, for example, PLOT3D contains a routine to locate and display three-dimensional shocks that have been captured by a numerical scheme. The experience is that strong shocks can be relatively easily identified, especially supersonic to subsonic flow shocks. Supersonic to supersonic shocks are more difficult to identify, especially as they are weak shocks. However, for a weak oblique shock and small flow deflection in two dimensions one obtains

$$\Delta p = -\rho_1 q_1 \Delta q$$

where q is the velocity and subscript 1 denotes conditions ahead of the shock. Thus it appears that if the interface boundary points are simply updated using the differenced governing equations, this will be adequate for weak shocks. Stronger shocks can be located and fit.

Mark Stewart, (Postdoctoral) Princeton University

The research project involves improving 2D multiblock grid generation and Euler equation solution techniques as well as their application to engine configurations. Previously, solutions to the Euler equations have been demonstrated for multi-element airfoils. With the objective of extending the techniques to engine configurations, geometry data for the Energy Efficient Engine (E^3) was obtained from General Electric Aircraft Engines. Grids and solutions for this case are currently being tested. Supporting this work are efforts to improve the domain decomposition techniques involved in the grid generation process.

Tim Swafford, Mississippi State University

Considerable progress concerning the aero-elastic analysis of the SR3 propfan was made during this two week period under the ICOMP program (June 18 through June 29, 1990) in that the merging of three-dimensional SR3 propfan blade deflection results from the NASTRAN structural analysis program into Whitfield's flux-difference-split Euler solver was accomplished. The approach was to extract blade deflection information under steady loading from NASTRAN and assume that these deflections represented the magnitude of a sinusoidal oscillation (at each blade surface grid point) at some pre-determined frequency. The problem then was to formulate a method by which the blades could be rotated and deflected while simultaneously deforming the computational grid commensurate with this motion. Unfortunately, it was determined that coordinate systems used in the fluid and structures analyses were not compatible due to the rotating grid being referenced to a stationary frame. Consequently, a considerable amount of time was spent

making the structural deflections ensuing from NASTRAn suitable for use in the flow solver routines concerned with grid deformation. This was accomplished and the resulting blade motion (with no rotation) was studied using animation techniques on an IRIS workstation. It now appears that unsteady flow field solutions can be obtained using these forced motion deflections at zero interblade phase angles. It should be noted that the assistance of Dennis L. Huff and T. S. Reddy of NASA Lewis (LeRC) was instrumental to the success of these efforts.

Eli Turkel, Tel Aviv University

Work continued with P. Jorgenson of NASA Lewis on the use of central difference schemes for time dependent problems with shocks. A TVD version of the central difference scheme with artificial viscosity was constructed by modifying the standard pressure switch. Other switches, based on minmod or superbee flux limiters, can also be constructed. The use of a matrix viscosity further increases the accuracy of the method. Numerous comparisons have been made on the 1D time dependent Euler equations.

Also, see BAYLISS, Alvin above.

William J. Usab, Jr., Purdue University

The objective of the present research is the development of an improved multigrid acceleration scheme for the solution of steady viscous flow problems. It has been observed that the application of current multigrid acceleration methods to problems involving highly stretched meshes with very large aspect ratio mesh cells results in very poor overall convergence rates. Since most Navier-Stokes codes rely on such meshes to efficiently resolve flow features within viscous shear layers the development of new multigrid acceleration methods which work well on such meshes remains a critical issue which must be addressed. The present research is being pursued with Dr. John Adamczyk and Mr. Mark Celestina at NASA Lewis.

To gain a better understanding of the origin of this convergence problem, the research during the present summer concentrated on determining how the "time" evolution of the discrete 2D solution is affected by the mesh stretching in one coordinate direction and local time-stepping. This was done by assuming an orthogonal mesh with uniform spacing in x and a variable, but prescribed spacing in y. Beginning with a standard multistage Runge-Kutta finite volume discrete formulation for the 2D euler equations with local time-stepping, the resulting equations were linearized, and Fourier-transformed in x. Then a Z-transform (Discrete Laplace transform) in time was used. These transformations reduce the problem to a system of discrete equations in η (transformed y-coordinate), for a given wave number in x, prescribed initial conditions and mesh stretching. While the resulting system of equations is still extremely difficult, if not impossible, to solve in closed form, if a particular mesh stretching is specified, one can look for approximate solutions or asymptotic behavior in the limit of high mesh stretching. Preliminary analysis for an exponentially stretched mesh suggests that for high mesh stretching with local time stepping, the x-dependence of the solution in the fine mesh region goes to zero. I am presently looking at possible ways to modify the present Runge-Kutta formulation which change the relative scaling of the terms in the transformed system of discrete equations as the solution evolves to the steady-state solution.

Also during this period, a new residual averaging operator based on a blend of second and fourth differences has been formulated. This new formulation has been incorporated into a 2D Euler/Navier-Stokes Runge-Kutta finite-volume code and is currently being tested. This research will be continued upon my return to Purdue.

Bram Van Leer, University of Michigan

Research was concentrated in two areas:

1. Preconditioning of the Euler equations by a local matrix.

   The six-wave model used previously was abandoned for lack of useful results. Starting with the case of 2D supersonic flow, the optimal matrix preconditioning was derived at the University of Michigan, in cooperation with Phil Roe; from this result the matrix for subsonic flow followed soon. The 3D form of the matrix was derived at ICOMP; the full results were presented in a seminar.

2. Flux-vector splitting without the calories.

The quest of a flux formula based on flux-vector splitting (FVS) that does not artificially increase transport of momentum and energy across a grid-aligned boundary layer was continued at the University of Michigan, in collaboration with Bill Coirier (NASA Lewis, presently a graduate student at the University of Michigan) and at ICOMP. A partial result, a mixture of FVS and flux-difference splitting (FDS), was reported at the NASA Lewis CFD Symposium on Aeropropulsion in April 1990; this idea was picked up by Liou and Steffen (NASA Lewis) and further improved.

The exploration of pure flux-vector splitting has lead to the conclusion that it is indeed possible to construct a FVS that does not diffuse boundary layers and is stable when used with explicit multi-stage time stepping. Robustness, though, is a problem; also, oscillatory behavior of the numerical solution near a shock has been observed. Presently, the splitting derived is, therefore, regarded as a mathematical exercise; greatest promise lies in the mixing of FDS and FVS.

Dave Whitfield, Mississippi State University

In efforts to solve the incompressible Euler or Navier-Stokes equations using a high resolution flux-difference split (Roe) implicit scheme, various approximations have been made for the Jacobian matrices used in the solution matrix. Because these approximations worked reasonably well for the solution of the incompressible equations, these same approximations were incorporated in the compressible version of the code. One reason for doing this, is to try and make the solution matrix more compatible with the right-hand side rather than the inconsistency used now due to the use of flux-vector splitting on the left-hand side and flux-differencing on the right-hand side. This approach of using approximations in compressible flow that are similar to those used in incompress-ible flow has not produced the improvements anticipated. However, during the course of this work, another approach involving direct numerical discretization of the flux vector was developed that made the left-hand side compatible with the right-hand side for both compressible and incompressible flow. The approach was coded in the compressible version and found to work reasonably well. It is currently being coded in the incompressible version.

Micha Wolfshtein, Technion, Israel Institute of Technology

The problem of turbulence modelling for compressible flows was considered. The approach chosen was to work on the Reynolds averaged equations, without mass averaging. To pursue this goal, it became necessary to use approximations to obtain an expression for the density-velocity correlations. The approach was based on the fluctuating equation of state which relates the fluctuating pressure, temperature and, density. This equation may be used to define various levels of approximation for the density fluctuations.

Some such possibilities have been examined. The first conceptually simple, but computationally very difficult possibility is to relate the density fluctuations to both the pressure and temperature fluctuations. The resulting equations are rather complicated, and require a lot of modelling. Another possibility is to assume that the density fluctuations in the energy carrying structures are isentropic about the mean density. This allows a prescription of the density fluctuations as a function of either temperature or pressure fluctuations. Yet this approach does not account for simultaneous changes in both the pressure and temperature which do not necessarily follow the isentropic equations. Yet, another possibility is to define approximate relations between the temperature and pressure fluctuations and the velocity fluctuations and find the relation between density and velocity fluctuations from these approximate relations. During the visit these approaches were examined. The final approach appears to offer more flexibility and has, therefore, some advantages over the first two approaches, although it is less rigorous.

Incorporation of the results in the convective terms of the compressible turbulent energy equation results in some additional terms involving high order moments of the velocity, like the turbulent energy, the Reynolds stresses, and triple and quadruple velocity correlations. Although such terms add complexity to the equation, many of them are terms included in the model anyway, and, therefore, they do not augment the modelling difficulties.

Yau Shu Wong, University of Alberta

The development of efficient numerical techniques for large nonsymmetric and indefinite systems of equations is of interest to scientific computations as well as of practical importance in engineering applications. Numerical simulations of many problems in fluid dynamics and aerodynamics by implicit (finite difference or finite element) methods often require the solution to a large set of linearized equations. The system is usually nonsymmetric and indefinite. Since the large system has to be solved repeatedly until a converged solution is obtained, the use of an efficient solver plays a major role in the success of a numerical simulation program.

Direct methods for solving such linear systems are not possible for large 2D and most 3D problems. Iterative methods used in conjunction with a suitable preconditioner have been very successful for symmetric and positive definite (SPD) matrix computations. However, numerical techniques for nonsymmetric and indefinite matrices have not been well-developed compared to those for the SPD problems. The conjugate gradient (CG) method has proven to be one of the most efficient iterative techniques for SPD matrices. In order to extend the application range to include the unsymmetric and indefinite systems, a conjugate-gradient-squared (CGS) method can be derived by squaring the basic CG algorithm. However, the CGS method has not been popular (even though the computational work per iteration and the computer storage requirement are more efficient than other iterative methods such as the GMRES), because the convergence behavior associated with the CGS method is somewhat erratic. An improved version of the CGS algorithm has been developed, and it has been tested for some nonsymmetric and indefinite systems. The improved CGS method is currently being incorporated with a preconditioning technique and its applications in implicit finite difference methods to the Euler equations will be investigated.

Zhigang Yang (Postdoctoral) Cornell University

Since joining LeRC/ICOMP in July 1990, I have been working on two research projects. The first project concerns the Renormalization Group (RNG) analysis of turbulence. The aim is to understand, validate the RNG approach in turbulence, and study the possibility of constructing a RNG based turbulence model. The second project is on the calculation of bypass transition based on turbulence modelling.

The writing of two papers on the work I did at Cornell was finalized at Lewis. The first paper entitled, "Nonlinear Dynamics Near the Stability Margin in Rotating Pipe Flow," was submitted to JFM for consideration of publication. The second paper entitled, "On the Stability of Viscous Wall Mode in Rotating Pipe Flow," was submitted for consideration of presentation in the coming AIAA 22nd Fluid Dynamics, Plasma Dynamics and Lasers Conference. Both papers are co-authored with Sid Leibovich of Cornell University.

Shaye Yungster, (Postdoctoral) University of Washington

The CFD research effort carried out at ICOMP during the last year has been aimed at studying the superdetonative modes of the ram accelerator, a scramjet-in-tube concept. These propulsion modes, which utilize oblique detonation waves or other shock-induced combustion modes, are similar to cycles proposed for NASP. The investigation has been carried out using a newly developed fully implicit, total variation diminishing (TVD) CFD code, that solves the fully coupled Reynolds-averaged Navier-Stokes equations and species continuity equations in an efficient manner. This code has been extensively tested against experimental and numerical results.

The first objective of the study was to obtain a fundamental understanding of the interactions that occur when a shock-wave impinges on a boundary layer consisting of premixed combustible gases. For this purpose, a computational study of shock-wave/boundary layer interactions in premixed hydrogen/air hypersonic flows was conducted. The results of this study are presented in ICOMP Report 90-22 (NASA TM-103273). This report describes the numerical scheme used, and contains also a study on the use of extrapolation techniques (i.e., reduced rank extrapolation and minimum polynomial extrapolation) for accelerating the convergence rate of the basic iterative scheme. Preliminary results show significant improvements in convergence rates for reacting and nonreacting flows, with very small increase in CPU time.

Investigation of complete ram accelerator configurations was also started. Preliminary estimates of the performance characteristic have been obtained at various Mach numbers. The results indicate a new combustion mechanism in which a shock wave induces combustion in the boundary layer. Two types of

solutions, one stable and the other unstable, have been observed so far, depending on the inflow conditions. In the latter case, a detonation wave is formed which propagates upstream and unstarts the diffuser. In the former case, a solution that converges to steady-state is obtained in which the combustion wave remains stationary with respect to the ram accelerator projectile. The possibility of stabilizing the detonation wave by means of a backward facing step is currently being investigated.

In addition to these studies, a new numerical scheme that eliminates the expense of inverting the large block matrices that arise in chemically reacting flows is currently being tested. In this formulation, the Jacobian matrix of the chemical source terms is replaced by a diagonal matrix designed to approximate the time scaling effects obtained with the Jacobian. This formulation has the potential to reduce significantly the computational work required to reach convergence, particularly when the number of chemical species considered is large.

REPORTS AND ABSTRACTS

The ICOMP Steering Committee: "Institute for Computational Mechanics in Propulsion (ICOMP), Fourth Annual Report — 1989," ICOMP Report No. 90-1, NASA TM-102519, March 1990, 58 pages.

The Institute for Computational Mechanics in Propulsion (ICOMP) is operated jointly by Case Western Reserve and the NASA Lewis Research Center in Cleveland, Ohio. The purpose of ICOMP is to develop techniques to improve problem-solving capabilities in all aspects of computational mechanics related to propulsion. This report describes the activities at ICOMP during 1989.

Hagstrom, Thomas (ICOMP): "Asymptotic Boundary Conditions for Dissipative Waves: General Theory," ICOMP Report No. 90-02, NASA TM-102497, January 1990, 20 pages.

An outstanding issue in the computational analysis of time dependent problems is the imposition of appropriate radiation boundary conditions at artificial boundaries. In this work we develop accurate conditions based on the asymptotic analysis of wave propagation over long ranges. Employing the method of steepest descents, we identify dominant wave groups and consider simple approximations to the dispersion relation in order to derive local boundary operators. The existence of a small number of dominant wave groups may be expected for systems with dissipation. Estimates of the error as a function of domain size are derived under hypotheses, leading to convergence results. Some practical aspects of the numerical construction of the asymptotic boundary operators are also discussed.

Lin, Avi (ICOMP): "Parallel Algorithms for Boundary Value Problems," ICOMP Report No. 90-03, NASA TM-102498, January 1990, 21 pages.

In the present paper we discuss a general approach to solve Boundary Value Problems numerically in a parallel environment. The basic algorithm consists of two steps: The local step where all the P available processors work in parallel, and the global step where one processor solves a tridiagonal linear system of the order P. The main advantages of this approach are two fold: First-this suggested approach is very flexible, especially in the local step and thus the algorithm can be used with any number of processors and with any of the SIMD or MIMD machines. Secondly-the communication complexity is very small and thus can be used as easily with shared memory machines. Several examples for using this strategy are discussed.

Kim, S.-W. (ICOMP): "Numerical Investigation of Separated Transonic Turbulent Flows With a Multiple-Time-Scale Turbulence Model," ICOMP Report No. 90-04, NASA TM-102499, January 1990, 40 pages.

Numerical investigation of transonic turbulent flows separated by streamline curvature and shock wave-boundary layer interaction is presented. The free stream Mach numbers considered are 0.4, 0.5, 0.6, 0.7, 0.8, 0.825, 0.85, 0.875, 0.90, and 0.925. In the numerical method, the conservation of mass equation is replaced by a pressure correction equation for compressible flows and thus incremental pressure is solved for instead of density. The turbulence is described by a multiple-time-scale turbulence model supplemented with a near-wall turbulence model. The present numerical results show that there exists a reversed flow region at all free stream Mach numbers considered whereas various k-_ turbulence models fail to predict such a reversed flow region at low free stream Mach numbers. The numerical results also show that the size of the reversed flow region grows extensively due to the shock wave-turbulent boundary layer interaction as the free stream Mach number is increased. These numerical results show that the turbulence model can resolve the turbulence field subjected to extra strains caused by the streamline curvature and the shock wave-turbulent boundary layer interaction and that the numerical method yields a significantly accurate solution for the complex compressible turbulent flow.

Hagstrom, Thomas (ICOMP): "Conditions at the Downstream Boundary for Simulations of Viscous Incompressible Flow," ICOMP Report No. 90-05, NASA TM-102510, February 1990, 29 pages.

The paper specification of boundary conditions at artificial boundaries for the simulation of time-dependent fluid flows has long been a matter of controversy. In this work we apply the general theory of asymptotic boundary conditions for dissipative waves, developed by the author in (8), to the design of simple, accurate conditions at a downstream boundary for incompressible flows. For Reynolds numbers far enough below the critical value for linear stability, a scaling is introduced which greatly simplifies the construction of the asymptotic conditions. Numerical experiments with the nonlinear dynamics of vortical disturbances to plane Poiseuille flow are presented which illustrate the accuracy of our approach. The consequences of directly applying the scalings to the equations are also considered.

Gajjar, J. S. B. (ICOMP): "Amplitude-Dependent Neutral Modes in Compressible Boundary Layer Flows," ICOMP Report No. 90-06, NASA TM-102524, March 1990, 18 pages.

The ideas of Benny & Bergeron (1969) and Davies (1970) on nonlinear critical layers are extended and some new nonlinear neutral modes are computed for compressible boundary layer flow. A special case of the work is when the generalized inflexion point criterion holds. Neutral modes are found for a range of phase-speeds, dependent on the Mach number, and the properties of these are discussed. As in the linear case when the flow is relatively supersonic, multiple neutral modes exist. The behavior of the neutral amplitude in some limiting cases is also considered, and it is found that the results are significantly different from that in incompressible flow when the flow is locally supersonic.

Duck, Peter W. (ICOMP): "Unsteady Three-Dimensional Marginal Separation, Including Breakdown," ICOMP Report No. 90-07, NASA TM-102525, February 1990, 30 pages.

We consider a situation involving a three-dimensional marginal separation, where a (steady) boundary-layer flow is on the verge of separating at a point (located along a line of symmetry/centreline). At this point we include a "triple-deck", thereby permitting a small amount of interaction to occur. Unsteadiness is included within this interaction region through some external means. It is shown that the problem reduces to the solution of a non-linear, unsteady, partial-integro system, which is solved numerically by means of time-marching together with a pseudo-spectral method spatially. A number of solutions to this sytem are presented which strongly suggest a breakdown of this system may occur, at a finite spatial position, at a finite time. The structure and details of this breakdown are then described.

Borettl, A. A. (ICOMP): "Three-Dimensional Euler Time Accurate Simulations of Fan Rotor-Stator Interactions," ICOMP Report No. 90-08, NASA TM-102528, December 1990, __ pages.

The paper presents a numerical method useful to describe unsteady 3D flow fields within turbomachinery stages. The method solves the compressible, time-dependent, Euler conservation equations with a finite-volume, flux-splitting, total-variation-diminishing, approximately-factored, implicit scheme. Multiblock composite gridding is used to partition the flow field into a specified arrangement of blocks with static and dynamic interfaces. The code is optimised to take full advantage of the processing power and speed of the Cray Y-MP supercomputer. The method is applied to the computation of the flow field within a single-stage, axial flow fan, thus reproducing the unsteady 3D rotor-stator interaction.

Leonard, B. P. (ICOMP), and Niknafs, H. S (The Norton Company): "A Cost-Effective Strategy for Nonoscillatory Convection Without Clipping," ICOMP Report No. 90-09, NASA TM-102538, March 1990, 24 pages.

Clipping of narrow extrema and distortion of smooth profiles is a well-known problem associated with so-called "high-resolution" nonoscillatory convection schemes. In this report, a strategy is presented for accurately simulating highly convective flows containing discontinuities such as density fronts or shockaves, without distorting smooth profiles or clipping narrow local extrema. The convection algorithm is based on non-artificially-diffusive third-order upwinding in smooth regions, with automatic adaptive stencil expansion to (in principle, arbitrarily) higher order upwinding locally, in regions of rapidly changing gradients. This is highly cost-effective because the wider stencil is used only where needed- in isolated narrow regions. A recently developed universal limiter assures sharp monotonic resolution of discontinuities without introducing artificial diffusion or numerical compression. An adaptive discriminator is constructed to distinguish between spurious overshoots and physical peaks; this automatically relaxes the limiter near local turning points, thereby avoiding loss of resolution in narrow extrema. Examples are given for one-dimensional pure convection of scalar profiles at constant velocity.

Gupta, Murli M. (ICOMP): "High Accuracy Solutions of Incompressible Navier-Stokes Equations," ICOMP Report No. 90-10, NASA TM-102539, March 1990, 26 pages.

In recent years we have developed high accuracy finite difference approximations for partial differential equations of elliptic type, with particular emphasis on the convection-diffusion equation. These approximations are of compact type, have a local truncation error of fourth order, and allow the use of standard iterative schemes to solve the resulting systems of algebraic equations. In this paper, we extend these high accuracy approximations to the solution of Navier-Stokes equations. Solutions are obtained for the model problem of driven cavity and are compared with solutions obtained using other approximations and those obtained by other authors. It is discovered that the high order approximations do indeed produce high accuracy solutions and have a potential for use in solving important problems of viscous fluid flows.

Mital, Subodh K. (ICOMP) and Chamis, Christos C. (NASA Lewis): "Fiber Pushout Test: A Three-Dimensional Finite Element Computational Simulation," ICOMP Report No. 90-11, NASA TM-102565, April 1990, 18 pages.

A fiber pushthrough process has been computationally simulated using three-dimensional finite element method. The interface material is replaced by an anisotropic material with greatly reduced shear modulus in order to simulate the fiber pushthrough process using a linear analysis. Such a procedure is easily implemented and is computationally very effective. It can be used to predict fiber pushthrough load for a composite system at any temperature. The average interface shear strength obtained from pushthrough load can easily be separated into its two components: one that comes from frictional stresses and the other that comes from chemical adhesion between fiber and the matrix and mechanical interlocking that develops due to shrinkage of the composite because of phase change during the processing. Step-by-step procedures are described to perform the computational simulation, to establish bounds on interfacial bond strength and to interpret interfacial bond quality.

Leonard, B. P. (ICOMP) and Mokhtari, Simin (University of Akron): "ULTRA-SHARP Nonoscillatory Convection Schemes for High-Speed Steady Multidimensional Flow," ICOMP Report No. 90-12, NASA TM-102568, April 1990, 50 pages.

For convection-dominated flows, classical second-order methods are notoriously oscillatory and often unstable. For this reason, many computational-fluid-dynamicists have adopted various forms of (inherently stable) first-order upwinding over the past few decades. Although it is now well known that first-order convection schemes suffer from serious inaccuracies attributable to artificial viscosity or numerical diffusion under high-convection conditions, these methods continue to enjoy widespread

popularity for numerical heat-transfer calculations, apparently due to a perceived lack of viable high-accuracy alternatives. But alternatives are available. For example, nonoscillatory methods used in gasdynamics, including currently popular "TVD" schemes, can be easily adapted to multidimensional incompressible flow and convective transport. This, in itself, would be a major advance for numerical convective heat transfer, for example. But, as this report shows, second-order TVD schemes form only a small, overly restrictive, subclass of a much more universal, and extremely simple, nonoscillatory flux-limiting strategy which can be applied to convection schemes of arbitrarily high-order accuracy, while requiring only a simple tridiagonal ADI line-solver, as used in the majority of general-purpose iterative codes for incompressible flow and numerical heat transfer. The new universal limiter and associated solution procedures form the so-called ULTRA-SHARP alternative for high-resolution nonoscillatory multidimensional steady-state high-speed convective modeling.

Michelassi, V. (ICOMP), Liou, M.-S. (NASA Lewis) and Povinelli, L.A. (NASA Lewis): "Implicit Solution of Three-Dimensional Internal Turbulent Flows," ICOMP Report No. 90-13, NASA TM-103099, July 1990, 44 pages.

The scalar form of the approximate factorization method has been used to develop a new code for the solution of three-dimensional internal laminar and turbulent compressible flows. The Navier-Stokes equations in their Reynolds-averaged form are iterated in time until a steady solution is reached. Evidence is given to the implicit and explicit artificial damping schemes that proved to be particularly efficient in speeding up convergence and enhancing the algorithm robustness. A conservative treatment of these terms at domain boundaries is proposed in order to avoid undesired mass and/or momentum artificial fluxes. Turbulence effects are accounted for by the zero-equation Baldwin-Lomax turbulence model and the q-ω two-equation model. For the first, an investigation on the model behavior in case of multiple boundaries is performed. The flow in a developing S-duct is then solved in the laminar regime at Reynolds number (Re) 790 and in the turbulent regime at Re = 40 000 using the Baldwin-Lomax model. The Stanitz elbow is then solved using an inviscid version of the same code at M_{inlet} = 0.4. Grid dependence and convergence rate are investigated showing that for this solver the implicit damping scheme may play a critical role for convergence characteristics. The same flow at Re = 2.5×10^6 is solved with the Baldwin-Lomax and the q-ω models. Both approaches showed satisfactory agreement with experiments, although the q-ω model is slightly more accurate.

Mital, Subodh K. (ICOMP), Caruso, John J. (NASA Lewis) and Chamis, Christos C. (NASA Lewis): "Metal Matrix Composites Microfracture: Computational Simulation," ICOMP Report No. 90-14, NASA TM-103153, November 1990, 14 pages.

Fiber/matrix fracture and fiber-matrix interface debonding in a metal matrix composite (MMC) are computationally simulated. These simulations are part of a research activity to develop computational methods for microfracture, microfracture propagation and fracture toughness of the metal matrix composites. The three-dimensional finite element model used in the simulation consists of a group of nine unidirectional fibers in three by three unit cell array of SiC/Ti15 metal matrix composite with a fiber volume ratio of 0.35. This computational procedure is used to predict the fracture process and establish the hierarchy of fracture modes based on strain energy release rate. It is also used to predict stress redistribution to surrounding matrix/fibers due to initial and progressive fracture of fiber/matrix and due to debonding of fiber-matrix interface. Microfracture results for various loading cases such as longitudinal, transverse, shear and bending are presented and discussed. Step-by-step procedures are outlined to evaluate composite microfracture for a given composite system.

Demuren, A. O. (ICOMP): "Calculation of 3D Turbulent Jets in Crossflow With a Multigrid Method and a Second-Moment Closure Model," ICOMP Report No. 90-15, NASA TM-103159, September 1990, 18 pages.

A multigrid method is presented for calculating turbulent jets in crossflow. Fairly rapid convergence is obtained with the k-ϵ turbulence model, but computations with a full Reynolds stress turbulence model

(RSM) are not yet very efficient. Grid dependency tests show that there are slight differences between results obtained on the two finest grid levels. Computations using the RSM are significantly different from those with k-ε model and compare better to experimental data. Some work is still required to improve the efficiency of the computations with the RSM.

Shih, T. H. (ICOMP): "An Improved k-ε Model for Near-Wall Turbulence and Comparison With Direct Numerical Simulation," ICOMP Report No. 90-16, NASA TM-103221, August 1990, 23 pages.

This paper presents an improved k-e model for low Reynolds number turbulence near a wall. The near-wall asymptotic behavior of the eddy viscosity and the pressure transport term in the turbulent kinetic energy equation is analyzed. Based on this analysis, a modified eddy viscosity model, having correct near-wall behavior, is suggested, and a model for the pressure transport term in the k-equation is proposed. In addition, a modeled dissipation rate equation is reformulated. We use fully developed channel flows for model testing. The calculations using various k-ε models are compared with direct numerical simulations. The results show that the present k-ε model performs well in predicting the behavior of near-wall turbulence. Significant improvement over previous k-ε models is obtained.

Shih, T. H. (ICOMP) and Mansour, N. N. (NASA Ames): "Modeling of Near-Wall Turbulence," ICOMP Report No. 90-17, NASA TM-103222, September 1990, 14 pages.

This paper presents an improved k-ε model and a second order closure model for low Reynolds number turublence near a wall. For the k-ε model, a modified form of the eddy viscosity having correct asymptotic near-wall behavior is suggested, and a model for the pressure diffusion term in the turbulent kinetic energy equation is proposed. For the second order closure model, we modify the existing models for the Reynolds-stress equations to have proper near-wall behavior. A dissipation rate equation for the turbulent kinetic energy is also reformulated. The proposed models satisfy realizability and will not produce unphysical behavior. Fully developed channel flows are used for model testing. The calculations are compared with direct numerical simulations. It is shown that the present models, both the k-ε model and the second order closure model, perform well in predicting the behavior of the near wall turbulence. Significant improvements over previous models are obtained.

Lin, Avi (ICOMP): "Parallel/Distributed Direct Method for Solving Linear Systems," ICOMP Report No. 90-18, NASA TM-103229, July 1990, 18 pages.

In this paper, a new family of parallel schemes for directly solving linear systems will be presented and analyzed. It is shown that these schemes exhibit a near optimal performance and enjoy several important features:

1. For large enough linear systems, the design of the appropriate parallel algorithm is insensitive to the number of processors as its performance grows monotonically with them.

2. It is especially good for large matrices, with dimensions large relative to the number of processors in the system. In this case, it achieves

    an optimal speed up.
    an optimal efficiency.
    a very low communication time complexity.

3. It can be used in both distributed parallel computing environments and tightly coupled parallel computing systems.

4. This set of algorithms can be mapped onto any parallel architecture without any major programming difficulties or algorithmical changes.

Hagstrom, Thomas (ICOMP): "Asymptotic Analysis of Dissipative Waves With Applications to Their Numerical Simulation," ICOMP Report No. 90-19, NASA TM-103231, August 1990, 13 pages.

This paper is concerned with various problems involving the interplay of asymptotics and numerics in the analysis of wave propagation in dissipative systems. A general approach to the asymptotic analysis of linear, dissipative waves is developed. We apply it to the derivation of asymptotic boundary conditions for numerical solutions on unbounded domains. Applications include the Navier-Stokes equations. Multidimensional traveling wave solutions to reaction-diffusion equations are also considered. We present a preliminary numerical investigation of a thermodiffusive model of flame propagation in a channel with heat loss at the walls.

Sidi, Avram (ICOMP): "Efficient Implementation of Minimal Polynomial and Reduced Rank Extrapolation Methods," ICOMP Report No. 90-20, NASA TM-103240, August 1990, 42 pages.

The minimal polynomial extrapolation (MPE) and reduced rank extrapolation (RRE) are two very effective techniques that have been used in accelerating the convergence of vector sequences, such as those that are obtained from iterative solution of linear and nonlinear systems of equations. Their definitions involve some linear least squares problems, and this causes difficulties in their numerical implementation. In this work timewise efficient and numerically stable implementations for MPE and RRE are developed. A computer program written in FORTRAN 77 is also appended and applied to some model problems.

Hagstrom, Thomas (ICOMP): "Consistency and Convergence for Numerical Radiation Conditions," ICOMP Report No. 90-21, NASA TM-103262, August 1990, 12 pages.

We consider the problem of imposing radiation conditions at artificial boundaries for the numerical simulation of wave propagation. Our emphasis is on the behavior and analysis of the error which results from the restriction of the domain. We give a brief outline of the theory of error estimation for boundary conditions. Use is made of the asymptotic analysis of propagating wave groups to derive and analyze boundary operators. For dissipative problems this leads to local, accurate conditions, but falls short in the hyperbolic case. A numerical experiment on the solution of the wave equation with cylindrical symmetry is described. We give a unified presentation of a number of conditions which have been proposed in the literature and display the time dependence of the error which results from their use. The results are in qualitative agreement with theoretical considerations. We find, however, that for this model problem it is particularly difficult to force the error to decay rapidly in time.

Yungster, Shaye (ICOMP): "Numerical Study of Shock-Wave/Boundary Layer Interactions in Premixed Hydrogen-Air Hypersonic Flows," ICOMP Report No. 90-22, NASA TM-103273, AIAA 91-0413, November 1990, 22 pages.

A computational study of shock wave/boundary layer interactions involving premixed combustible gases, and the resulting combustion processes is presented. The analysis is carried out using a new fully implicit, total variation diminishing (TVD) code developed for solving the fully coupled Reynolds-averaged Navier-Stokes equations and species continuity equations in an efficient manner. To accelerate the convergence of the basic iterative procedure,, this code is combined with vector extrapolation methods. The chemical nonequilibrium processes are simulated by means of a finite-rate chemistry model for hydrogen-air combustion. Several validation test cases are presented and the

results compared with experimental data or with other computational results. The code is then applied to study shock-wave/boundary layer interactions in a ram accelerator configuration. Results indicate a new combustion mechanism in which a shock wave induces combustion in the boundary layer, which then propagates outwards and downstream. At higher Mach numbers, spontaneous ignition in part of the boundary layer is observed, which eventually extends along the entire boundary layer at still higher values of the Mach number.

Arnone, A. (ICOMP), LIOU, M.-S. (NASA Lewis) and Povinelli, L.A. (NASA Lewis): "Navier-Stokes Analysis of Transonic Cascade Flow," ICOMP Report No. 90-23, NASA TM-103624, October 1990, 11 pages.

A new kind of C-type grid is proposed, this grid is non-periodic on the wake and allows minimum skewness for cascades with high turning and large camber. Reynolds-averaged Navier-Stokes equations are solved on this type of grid using a finite volume discretization and a full multigrid method which used Runge-Kutta stepping as driving scheme. The Baldwin-Lomax eddy-viscosity model is used for turbulence closure. A detailed numerical study is proposed for a highly loades transonic blade. A grid independence analysis is presented in terms of pressure distribution, exit flow angles, and loss coefficient. Comparison with experiments clearly demonstrates the capability of the proposed procedure.

Bayliss, Alvin (ICOMP) and Turkel, Eli, (ICOMP): "Mappings and Accuracy for Chebyshev Pseudo-spectral Approximations," ICOMP Report No. 90-24, NASA TM-103630, December 1990, 20 pages.

The effect of mappings on the approximation, by Chebyshev collocation, of functions which exhibit localized regions of rapid variation is studied. A general strategy is introduced whereby mappings are adaptively constructed which map specified classes of rapidly varying functions into low order polynomials which can be accurately approximated by Chebyshev polynomian expansions. A particular family of mappings constructed in this way is tested on a variety of rapidly varying functions similar to those occurring in approximations. It is shown that the mapped function can be approximated much more accurately by Chebyshev polynomial approximation than in physical space or where mappings constructed from other strategies are employed.

Goldstein, M. E. (NASA Lewis), Leib, S. J. (Sverdrup), and Cowley, S. J. (ICOMP): "The Effect of Small Streamwise Velocity Distortion on the Boundary Layer Flow Over a Thin Flat Plate With Application to Boundary Layer Stability Theory," ICOMP Report No. 90-25, NASA TM-103668, December 1990, 36 pages.

We show how an initially linear spanwise disturbance of the free-stream velocity field is amplified by leading edge bluntness effects and ultimately leads to a small amplitude but nonlinear spanwise motion far downstream from the edge. This spanwise motion is imposed on the boundary layer flow and ultimately causes an order-one change in its profile shape. The modified profiles are highly unstable and can support Tollmein-Schlichting wave growth well upstream of the theoretical lower branch of the neutral stability curve for a Blasius boundary layer.

31

SEMINARS

(* = CMOTT Seminars)

Afolabi, Dare (Purdue Unversity): "Modal Interaction in Dynamic Systems"

In structural dynamics, a complex engineering system is often analyzed as an assembly of coupled sub-structures. Under certain circumstances, "double modes" are encountered. We discuss the dynamics of interaction in such systems using modal analysis and singularity theory. The mechanics of energy interchange is examined, in light of the notion of "conversative coupling" recently introduced by Crandall. Application areas include bladed disks, propfans, engine/airframe interaction, tube partially filled with a fluid, etc.

Akl, Fred A. (Louisiana Tech University): "Parallel Eigen Problem Solver Using a Binary Three Transputer Network"

The presentation deals with the implementation of a parallel eigen problem solver on a network of up to 31 transputers configured in a binary tree topology. The method of recursive doubling due to Kogge (1974) will be briefly discussed and its application in implementing the eigen solver will be highlighted. Replication of the algorithm on a number of typical nodes and leafs permits flexibility in tailoring the algorithm to the depth of the binary tree. Performance of the algorithm on binary trees of depth 1, 2, 3, and 4 will be given.

Ameri, A. A. (Cleveland State University): Navier-Stokes Analysis of Turbomachinery Blade External Heat Transfer"

The two-dimensional, compressible, thin layer Navier-Stokes and energy equations were solved numerically to obtain heat transfer rates on turbomachinery blades. The Baldwin-Lomax algebraic model and the q-ω Low Reynolds Number Two-Equation model were used for modeling of turbulence. For the numerical solutions of the governing equations a four stage Runge-Kutta solver was employed. The turbulence model equations were solved using an implicit scheme. Numerical solutions are presented for two-dimensional flow over a flat plate cascade and three vane cascades. The heat transfer results and the pressure distributions were compared with published theoretical results and experimental data.

Balsa, Thomas F. (University of Arizona): "The Evolution of Nonlinear Disturb-ances in a Supersonic Mixing Layer"

High Mach number supersonic mixing layers are weakly unstable at low frequencies in the sense that the linear growth rate of a disturbance is small compared to its wavenumber. It is, therefore, possible to derive a nonlinear theory (for the evolution of a disturbance) which is obtained by the scaling ϵ (=characteristic amplitude) = σ (=characteristic frequency) << 1. The corresponding transverse structure of the flow field is described by three layers: the most important of which is the critical layer where $y/\epsilon = 0(1)$. Here the advection of the perturbation quantities (vorticity, temperature, etc.) depends on the velocity induced by the external disturbance (in the main part of the shear layer), whose amplitude is determined by a jump condition across the critical layer. The flow is fully nonlinear in the critical layer in the sense that all harmonics of the fundamental are present. The theory and some numerical results will be discussed.

Braaten, Mark E. (General Electric Research & Development Ctr.): "CFD On Distributed Memory MIMD Machines: Some Success, Some Frustration, Much Promise"

A parallel algorithm for the solution of the 2D compressible and incompressible Navier-Stokes equations has been developed and demonstrated on distributed memory multi-computers. The algorithm represents a

parallel implementation of an extended pressure correction formulation that simultaneously updates both velocities and density to enforce continuity, and upwinds density to allow shock capturing. This formulation is capable in principle of analyzing the flow in all of the major components of gas turbine engines, including inlet ducts, fans, compressors, combustors, turbines, and augmenters.

The parallel algorithm is based on a multi-dimensional domain decomposition. Results have been achieved on a first generation Intel iPSC/1 hypercube, and a second genera-tion Intel iPSC/2VX hypercube with both scalar and vector processors. Performance approaching 1/4 of a single Cray Y-MP processor has been achieved on 32 vector nodes. Currently, work is underway to implement the algorithm on a third generation iPSC/860 machine, and to develop a parallel 3D algorithm.

The talk will focus on the results obtained to date, comment on some of the problems encountered along the way, and discuss the potential of parallel computing for CFD.

Chen, J.-Y. (Sandia National Labs): "Monte Carlo Simulation of Turbulent React-ing Flows"

Monte Carlo simulation of joint scalar probability density function (pdf) is explored for modeling turbulent reacting flows with realistic chemistry. The general principles of stochastic simulation for turbulent convection, molecular mixing, and chemical reactions are discussed. Fundamental issues and practical limitations of the pdf methods are addressed. Realistic models for complex chemical reactions are developed and implemented in the stochastic simulations. Results demonstrate the ability of pdf method in predicting significantly nonequilibrium chemical processes such as local extinction and flame blow-out. Also indicated are the needs of a better molecular mixing model and a stochastic simulation for the interactions between turbulence and chemical reactions.

Chen, Kuo-Huey (Iowa State University): "A Primitive Variable, Strongly Implicit Calculation Procedure for Two and Three Dimensional Viscous Flows
Including Low Mach Number Compressible, Supersonic and Incompressible Liquid Sloshing Flows"

PART A: A Primitive Variable, Strongly Implicit Calculation Procedure for Viscous Flows at All Speeds

A coupled solution procedure is described for solving the compressible form of the time-dependent, two-dimensional Navier-Stokes equations in body-fitted curvilinear coordinates. This approach employs the strong conservative form of the governing equations but uses primitive variables (u, v, p, T) rather than the more traditional conservative variables (p, pu, pv, e_t) as unknowns. A coupled modified strongly implicit procedure (CMSIP) is used to efficiently solve the Newton-linearized algebraic equations. It appears that this procedure is effective for Mach numbers ranging from the incompressible limit to supersonic. Generally, smoothing was not needed to control spatial oscillations in pressure for subsonic flows despite the use of central differences. Dual-time stepping was found to further accelerate convergence for steady flows. Sample calculations, including steady and unsteady low Mach number internal and external flows and a steady shock-boundary layer interaction flow, illustrate the capability of the present solution algorithm.

PART B: Simulation of Liquid Sloshing in a Partially Filled Spherical Container

The same numerical solution procedure was used to solve three-dimensional liquid sloshing motion in a partially filled spherical container mounted on a spin-stabilized satellite. Several coordinate transformations for the governing equations were performed in order to follow the complicated rotation/nutation motion of the container and the free surface motion. Results will be presented for three different kinds of spin-up problems, including an animation of one of the results.

* **Claus, Russell (NASA Lewis):** "Chaotic Wanderings Through a Land of Turbulence Models"

A turbulence model user details his purusit of the mythological "correct answer." The talk opens with some failed attempts to calculate separated flows using a two equation turbulence model. Following this, additional attempts to achieve the "correct answer" through a series of Direct Numerical Simulations and Large Eddy

Simulatins will be described and the limitations of these approaches are highlighted. Finally, a re-examination of two-equation and a second order closure calculations of a jet in crossflow will be discussed.

A brief discussion of some turbulence modeling efforts being supported under the NASA SBIR program will also be described. This includes RNG modeling with Orszag and Yakhot, and PDF modeling for compressible flows with Kollman and Farshchi.

Demuren, A. O. (University of Lagos): "Calculation Of Complex 3D Turbulent Flows With a Second-Moment Closure"

A full Reynolds stress turbulence closure model is applied to calculate a range of complex turbulent flows. Partial differential equations are solved for the pressure, 3 components of the velocity vector, 6 components of the Reynolds stress tensor and a length scale using a finite-volume numerical procedure. For flows with heat transfer, additional differential equations are solved for the temperature and 3 components of the turbulent heat flux. Contrary to previous practice, no staggering of the velocity and Reynolds stress computational grids is necessary for stability of the numerical procedure. A multigrid scheme is utilized to accelerate the convergence of the numerical method. This enables the use of simple solvers with low operation count, which are also largely vectorizable. Computed results are compared with experimental data and results from eddy-viscosity based models, such as the k-ϵ model and extensions, for complex channel flows and for jets in crossflow.

de Vahl Davis, Graham (University of New South Wales): "A Numerical Study of Three Dimensional Natural Convection During the Freezing of Water"

A numerical study of three-dimensional natural convection with phase change in a rectangular parallepiped will be described. The fluid may have temperature-dependent properties. The governing equations have been approximated by finite differences, second order in space and first order in time. A boundary fitted coordinate system has been used to predict the shape and movement of the solid-liquid interface through time.

Preliminary results will be presented for the freezing of water in a cube for Rayleigh numbers in the range of about 10**5 - 10**6.

* Gibson, M. M. (Imperial College of Science Technology): "Pressure-Strain Modelling and the Return to Isotropy"

Calculations of complex shear layers give the best results when the constant in Rotta's return-to-isotropy model is given values greater than those deduced from the measurements in grid turbulence. New data from grid turbulence are presented and assessed in the light of previous experiments. The results show that homogeneous turbulence decay is associated with increasing Reynolds number. It is argued that this finding has important implications for Reynolds-stress Modelling. Recent theoretical and experimental studies of the analogous model for pressure scrambling in the scalar-flux equations reveal even larger discrepancies in them "Monin constant." The implications for the future of second-moment modelling are discussed.

Hagstrom, Thomas (State University of New York): "Error Estimation and Convergence for Numerical Radiation Boundary Conditions"

The problem of specifying accurate radiation boundary conditions is of importance for the simulation of many time dependent fluid flows and has been the subject of much research in recent years. Nonetheless, the basic issues of numerical analysis, error estimation and convergence, have in general not been addressed. In this talk we discuss the ingredients of the theory of these approximations. A general procedure for dissipative problems is developed which leads to convergence results. We demonstrate its effectiveness when applied to the Navier-Stokes equations. Accuracy for hyperbolic problems is more difficult to establish. We consider the simple case of the wave equation with cylindrical symmetry. A unified discussion of a number of

procedures from the literature is given. A numerical experiment is made to assess the accuracy of each approach, in particular for computations over long time. The potential application of these ideas to the Euler equations is also discussed.

Hariharan, S. I. (University of Akron): "Unresolved Aspects of Compressible External Unsteady Flows"

In this talk we consider computational issues around external compressible flows in two dimensions. The time accuracy of the solutions will be the central issue. Recent treatment of these problems and their deficiencies in the accuracy of numerical solutions over a long time will be shown. Our study shows that accurate modelling of the far field conditions is a major issue. We present a more general class of far field conditions which encompass both the low and high frequency behavior of the exact boundary operator. Of particular interest for problems in two dimensions is the use of an operator with nonlocal couplings in both space and time. Inclusion of these nonlocal couplings increases the computational time by orders of magnitude but is inevitable if one desires long time simulations of the results.

* Hirsch, Charles (Free University of Brussels): "CFD-Related Research at the University of Brussel and Turbulence Research Activities in Europe"

A review of our research will be presented.

* Lang, Nancy (NASA Lewis Intern): "A Comparative Analysis of Two Equation Turbulence Models"

Several two equation models have been proposed and tested against benchmark flows by various researchers. For each study, different numerical methods or codes were used to obtain the results which were an improvement or success over some other model. However, these comparisons may be overshadowed by the different numerical schemes used to obtain the results. With this in mind, several existing two equation turbulence models, including k-ϵ and k-τ models, are implemented into a common flow solver code for near wall turbulent flows. Calculations are carried out for low Reynolds number, two-dimensional, fully developed channel and boundary layer flows. The accuracy of the different models is established by comparing the turbulent kinetic energy, mean velocity, and shear stress profiles with the direct numerical simulations and experimental data.

Leonard, B. P. (University of Akron): "Limited Higher Order Upwinding
(How to Avoid the "Wiggle-or-Smear" Compromise)"

In a well-known comparative study on high-convection techniques, Smith and Hutton concluded that "... there seems to be no best method ... only a compromise between wiggling and smearing ..." One might add "... and steepening-and-clipping ..." All these difficulties stem from trying to construct convection schemes from combinations of first- and second-order methods. If one instead uses third-or higher-order upwind interpolation, modified by simple limiter constraints, much better (sharp, nonoscillatory) results can be obtained. In addition to the Smith-Hutton and other scalar benchmark problems, results will be shown for a coarse-grid Navier-Stokes calculation (driven cavity), the one-dimensional Burgers equation, and transonic nozzle flow.

Lin, Avi (Temple University): "High Order Parallel Numerical Schemes for Fast Simulation of the Second Order Length Scale Turbulence Model"

The Turbulence Length Scale (TLS) model has been developed and enhanced under a NASA grant in the last two years. It closes the mean Navier-Stokes equations in the Reynolds Stress (RS) model. The closure model is based on the fact that the major terms in the RS equations (the pressure-interaction term and the

dissipation tensor) depend on the two-point velocity correlation. An integral form of the correlations define the TLS. The final system of dynamical equations is composed of the six RS equations and the six TLS equations.

The ongoing effort on the parallel numerical simulation of turbulent flows using the above approach is described. It is conducted in a cooperative effort between ICOMP and NASA Internal Fluid Mechanics Division. The emphasis is on accurate and fast solutions of internal flows. High order convection-diffusion schemes have been designed, analyzed and programmed and were applied for all the dynamical equations under consideration. The flow is solved in primitive variables, where the suggested parallel numerical scheme is unconditionally stable, with enough parameters to maximize the rate of convergence.

. Several parallel algorithms of different parallel granularity have been developed and programmed, and serve as the major building blocks in using the parallel environment for these internal flow equations. The Hypercluster hardware environment has been used to study the feasibility of this approach.

In addition to the above issues, and presenting some (parallel computing) results for sample of flows, the pros and cons of this approach will be assessed.

Lumley, John L. (Cornell University): "Measurements in a Circular Jet of Helium into Quiescent Air*"

The ability to model an inert flow with density fluctuations allows one to model a diffusion flame with fast chemistry, and gives insight into one aspect of the dynamics of compressible flows. With this in view, these measurements were made to calibrate a second order model for such flows. Measurements using shuttle-mounted hotwire and Way-Libby probes are described. For comparison, a jet of air into air was also measured. Moments up to fourth order were computed. The data are used to evaluate various modeling assumptions. Attempts are made to explain the greater spreading rate of the helium jet.

* From the Ph.D. thesis of N. R. Panchapakesan

Mawid, Mohammd A. (University of Illinois-Chicago): "Numerical Simulation Of a Laminar Diffusion Flame Supported by Liquid Fuel Droplet Streams"

Some computations of a laminar diffusion flame supported by droplet streams have been performed using the KIVA-II computer code. The difficulty associated with numerical simulations of two-phase laminar reactive flows is due to the exchange process between the gas and liquid phases. To accurately solve for the essential dynamics of the injected liquid fuel and its interactions with the gas-phase, detailed droplet heating models to represent the transient heat diffusion within each droplet, stochastic injection models and improved droplet vaporization models are all required. In addition, either detailed or fast-chemistry (nonequilibrium and equilibrium chemistry), depending upon the chemical reaction time scale as compared to diffusive convective time scales, is also essential.

The diffusion-limit model to simulate the transient heat transport within each injected liquid droplet (or groups of droplets) and an n-injector element option to represent the injection of liquid fuel streams have been implemented so far in KIVA-II. The numerical results obtained so far show that an underventilated diffusion flame emerges. The results also indicate that the flame structure somewhat resembles cellular flames in its motion. Further computations to show the effects of the various important parameters on the flame development and structure are being performed.

Messiter, A. F. (University of Michigan): "Hypersonic Boundary-Layer Blowing"

Asymptotic solutions are derived for distributed or strip blowing from a flat plate or a wedge at high Mach number. The blowing is strong enough that the boundary layer is blown away from the surface as a high-temperature, free shear layer, here assumed laminar. For power-law blowing from a cooled wall, self-similar solutions are found in each of three "layers": 1) an inviscid outer layer, 2) an inviscid blown-gas layer, and 3) the viscous mixing layer. For strip blowing, separation occurs ahead of blowing and is described by a "triple-deck" hypersonic free interaction; pressures are then derived in the blowing region.

* Michelassi, Vittorio (University of Florence): "Computation of Elliptic Flows Using Low-Reynolds-Number Two-Equation Models"

Two new low-Reynolds number forms of the k-ε model will be presented. These exhibit better stability and stiffness characteristics as compared to the previous formulations. Model generality is improved by formulating the damping functions so that they do not depend on the wall distance. The proposed formulations are compared with eight other low Reynolds number two-equation turbulence models by computing the fully developed channel flow and the incompressible flow past a hill. Results are compared with available direct numerical simulation and experimental data. The flow solver is based on the approximate factorization technique and the artificial compressibility method requiring no (or very little) numerical damping. A simple linearization technique for the turbulence model source terms based on Taylor series expansion ensures implicit algorithm stability for all the models tested. Both numerical accuracy and computational efficiency are discussed.

* Murthy, S. N. B. (Purdue University): "Heat Transfer with Inhomogeneous Free Stream Turbulence"

The presence of free stream turbulence (FST) in a wall-bounded flow with heat transfer presents several interaction complexities. These depend upon (a) the state of the boundary layer, laminar, fully turbulent or transitional, (b) the nature of FST, especially its "peakiness" or inhomogeneity, and (c) other complications such as pressure gradient, geometry and cooling. An investigation is in progress on the possible application of (a) large eddy interaction hypothesis (based on Lumley's rational description of turbulence) and (b) spectral analogy between heat and turbulence kinetic energy to the simplest case of a flat wall, fully developed turbulent boundary layer with zero pressure gradient when there is heat transfer in the presence of homogeneous FST. An extention of the approach to the case of other types of boundary layers with inhomogeneous FST will be discussed.

Pierre, Christophe (University of Michigan): "Comments on the Dynamics of Mistuned Blade Assemblies"

We discuss some aspects of the effects of random mistuning on the dynamics of turbomachinery rotors. On simple models, we illustrate the drastic phenomena of localization of the mode shapes and curve veering of the free vibration eigenvalues. We discuss the passband/stopband structure of periodic assemblies from a traveling wave viewpoint, and relate it to the modal description of the system. This allows us to introduce a probabilistic characterization of the degree of localization in the assembly — the so-called localization factor. We also develop a second-order stochastic measure of sensitivity to disorder that, contrary to first-order techniques, is able to capture large mistuning effects. We present preliminary results illustrating the increase in forced response amplitude due to mistuning, for a simple probabilistic model. Finally, we outline directions of future research on the forced response of mistuned assemblies.

Pletcher, Richard H. (Iowa State University): "Numerical Simulation Of Unsteady Viscous Flows"

Some recent results in the simulation of unsteady viscous flows will be presented. These will be drawn mainly from two studies. In the first, computations from two explicit and two implicit algorithms are compared with available analytical, computational, and experimental results for the oscillating flat plate and the impulsively started cylinder. The long term objective of the second study has been to develop a computational strategy that is efficient and accurate from the incompressible limit up through transonic speeds. The scheme used the strong conservation-law form of the equations, but with primitive rather than conserved variables. A coupled, strongly implicit scheme is used to solve the governing equations. Results will be shown for the unsteady vortex shedding flow over a circular cylinder, a steady shock-boundary layer interaction flow on a flat plate in a supersonic stream, the three-dimensional driven cavity, and the unsteady rotating flow in a spherical tank with a free surface.

* Rice, Edward J. (NASA Lewis): "Fundamental In-house Experiments to Support the Lewis Turbulence Modeling Program"

The fundamental experiments conducted by members of the Inlet, Duct, and Nozzle Flow Physics Branch of IFMD have been primarily for support of the shear flow control effort within the Division. Although this support effort is expected to continue, some work could be diverted (and in time expanded) to support the Turbulence Modeling Program currently underway within ICOMP. This new emphasis can be accomplished through an interactive effort between the Numerical Analysts and the Experimentalists. The current and planned experiments will be summarized in this talk and include: unsteady flow around airfoils (stationary and oscillating), 2D rapid diffusers (backward facing ramp), aspirated backward facing step, circular and rectangular jets (subsonic and supersonic), dual stream supersonic shear layer (annular geometry), and boundary layer transition. A swirl generator within the plenum of the CW-17, ERB rig allows the addition of swirling flow to any of the jet or shear layer experiments. The available and planned experimental instrumentation include: single, X-wire and three-wire hot wire anemometry, single and two element corona probe, multiple microphone and pressure transducer channels, Schlieren and laser-sheet flow visualization, and conventional and fiber-optic two-component LDA systems. The new 16 channel anemometry allows simultaneous measurement with 8 X-wires. Some sample data will be presented to illustrate the current capability.

The presentation will take 30 minutes and will be followed by 30 minutes of discussion between the numerical and experimental participants. The presentation will be general in nature intended mainly to acquaint the numerical analyst with the experimental capability which can support the numerical program through a cooperative effort.

Rodi, W. (University of Karlsruhe): "Current Trends in Turbulence Modelling"

A brief review is given of recent work in the area of modelling turbulence in near-wall regions and by Reynolds-stress-equation models. Various low-Reynolds-number versions of the k-ϵ model and their damping functions are examined with the aid of results from direct numerical simulations and they are compared with respect to their performance in calculating boundary layers under adverse and favorable pressure gradients. A two-layer model is presented in which near-wall regions are resolved with a one-equation model and the core region with the standard k-ϵ model. Various applications of this model are shown. The ability of the various models to simulate laminar-turbulent transition in boundary layers is discussed. Recent applications of a basic Reynolds-stress-equation model to complex flows of practical interest are presented. Finally, some recent proposals for improved Reynolds-stress-equation models are outlined and an outlook on possible future turbulence model developments is given.

Scheuerer, Georg (Gesellschaft fur Reaktorsicherheit mbh): "Finite-Volume Calculations of Separated Turbulent Flows Using Second-Moment Closure and Co-located Variable Arrangement"

A finite-volume calculation procedure using co-located (non-staggered) variable arrangement and second-moment turbulence closure is presented. Special interpolation procedures for cell-face velocities and Reynolds stresses are employed to prevent oscillatory solutions which might otherwise develop because of the co-located variable arrangement. Calculations for backward facing step obstacle flows are performed on three successively refined numerical grids. Solution errors are then estimated using a technique based on Richardson extrapolation. Agreement with available data is improved as compared to calculations using the k-ϵ eddy viscosity turbulence model. However, there are also shortcomings of the second moment turbulence closure, namely the prediction of the normal stresses in the separation zone due to inadequate modelling of the pressure strain wall-reflection terms and the shape of the velocity profiles in the reattachment zone. The seminar closes with an outlook to planned improvements of the second moment closure model.

Shih, T. H. (Stanford University-Ctr): "A Research Program for Turbulence Modeling in CFD"

1. Importance of turbulence modeling in CFD.

2. What does CFD need form turbulence modeling?

3. Various turbulence models and their limitations.
   One-point closure models.
   Two-point closure models.
   Numerical simulations.
   Other turbulent theories.

4. Current status of turbulence modeling.

5. Research proposal for turbulence modeling at LeRC.

* Shih, T. H. (Stanford University-Ctr): "Engineering Turbulence Modeling — Present and Future"

A summary of the present position of eddy-viscosity models (e.g. k-ε) and second-order closure models (Reynolds stress models) is presented. Typical examples (comparisons between model predictions and experiments) show their abilities as well as their limitations.

Development of more advanced and complex schemes is discussed. The inclusion of such models into a CFD commercial code is feasible, but need intensive work — a cooperative effort!

Sidi, Avram (Technion-Israel Institute of Technology): "Vector Valued Rational Approximations and Their Application to the Algebraic Eigenvalue Problem and Time-Periodic Steady-State Solutions of Continuum Problems"

We develop three rational approximation procedures for vector valued functions analytic at the origin and meromorphic (having only polar singularities) in a neighborhood of the origin. We state some convergence results for these procedures. Noting the analogy between such functions and the algebraic Eigenvalue problem and the time-periodic steady-state solutions of continuum problems, we propose new acceleration methods for the latter. Numerical examples of application to the algebraic Eigenvalue problem will be demonstrated.

Steger, Joseph L. (University of California-Davis): "On the Chimera Method of Simulation of Three-Dimensional Flow"

The chimera overset grid method for simulating flow about complex configurations will be reviewed. It is argued that the technique is competitive to other schemes and should ultimately prove more useful. Known deficiencies of the chimera approach will be discussed along with possible refinements.

Stewart, Mark (Princeton University): "Generation of Two-Dimensional Non-Overlapping Structured Grids for Multiple Element Airfoils"

Grid generation for numerical solutions to partial differential equations has limited the engineering simulations which can be done in geometrically complicated physical problems. There are many approaches

to grid generation including multiblock grids where the region is subdivided into topologically rectangular regions containing structured grids. However, the domain decomposition is generated manually.

A novel algorithm for substantially automating the decomposition of arbitrary, two-dimensional domains into nonoverlapping topologically rectangular blocks will be described. To demonstrate this technique, solutions to the Euler equations for several multielement airfoil sections will be presented, including a four-element landing configuration for the A310 Airbus. Further, accuracy estimates for the grids have been developed analytically, and approximated numerically to demonstrate the accuracy characteristics of the grids.

Tadjfar, Mehran (Ohio State University): "Receptivity Interaction of a Three-Dimensional Roughness Element with Time-Harmonic Free-Stream Disturbances"

Receptivity of a laminar boundary layer due to the interaction of time-harmonic, free-stream disturbances with a 3D roughness element is studied. It is shown that at high Reynolds numbers, the governing equations for the unsteady motion are the unsteady, linearized, 3D, triple-deck equations. These equations can only be solved numerically. In the absence of any roughness element, the free-stream disturbances, to the first order, produce the classical Stokes flow, in the thin Stokes layer near the wall (on the order of our lower deck). However, with the introduction of a small, 3D roughness element, the interaction between the hump and the Stokes flow introduces a spectrum of all spatial disturbances inside the boundary layer. For super-critical values of the scaled Strouhal number, $S_0 > 2.29 \ldots$, these T-S waves are amplified in a wedge-shaped region extending downstream of the hump.

Turkel, Eli (University of Tel Aviv): "Multigrid and Central Differences For Hypersonics"

We consider the use of multigrid to accelerate the convergence of a central difference scheme with artificial viscosity and a Runge-Kutta smoother for the fluid dynamic equations. We present some theoretical results on the multigrid for hyperbolic equations. We then demonstrate some of the changes that are needed to the standard Runge-Kutta codes for them to work for higher speeds. All solutions are second order accurate in space.

Results will be shown for Mach 10 and 20 turbulent flows about a NACA-0012 and a 3D Mach 6 flow about a bi-cone. Both convergence rates and accuracy of the solutions will be presented.

van Leer, Bram (University of Michigan): "Characteristic Time-Stepping"

Explicit marching methods used in Euler codes have largely been based on the technique of "local time-stepping," according to which each computational cell of fluid is made to evolve with the largest time-step allowed by stability considerations. Mathematically speaking, this corresponds to removing the stiffness due to spatial variations of the flow variables and the grid parameters by a local preconditioning of the residual of the equations by a <u>scalar</u> factor. Since the flow of a compressible fluid is described by a system of equations, a local matrix preconditioner would be more appropriate. For discrete approximations of the Euler equations this should remove the stiffness due to the differences in the propagation speeds of the various wave modes in the fluid. Analogously to local time-stepping, this could be called "characteristic time-stepping."

For 1D inviscid flow, the construction of an effective preconditioner is relatively simple, because the wave speeds and directions are known. In multidimensional flow, the situation is much complicated by the omni-directional propagation of acoustic information: there is no unique decomposition of the flow field in terms of discrete waves.

A new result for the 2D Euler equations will be presented in the form of a local matrix that reduces the condition number of the characteristic speeds to one in supersonic flow and to $(1-M^2)^{-1/2}$ in subsonic flow. The first numerical results confirm the expected benefit of this preconditioning to both single-grid and multi-grid calculations.

40

* Wolfshteln, Micha (Technion, Israel Institute of Technology): "On Length Scale Equations in Two-Equation Turbulence Models"

Two-equation turbulence models show a great survivability between the very successful mixing length models and the Reynolds stress models. Yet the models are often criticized, mainly on the wide scatter of the computed results and on the difficulties encountered in the derivation of a reliable scale equation. The problem has been difficult to resolve due to: (i) The large resources required for developing solvers and to run test cases on computers; (ii) The theoretical difficulties to derive turbulence models from the Navier-Stokes equations. The demands from a "good" turbulence model are difficult to satisfy, and are often conflicting with one another. This point will be illustrated in a discussion of some possible approaches to this problem. In particular we shall refer to well established models like the dissipation or length scale models, as well as newer models like the volume of turbulence or time scale models. A generalized two-equation turbulence model will be used to demonstrate a possible approach for the improvement of two-equation models. Finally, a fourth order boundary layer solver for the generalized two equations model will be presented. The solver can handle both compressible and incompressible flows, with any two-equation model, with or without wall functions. Some results will be presented for a flat plate boundary layer and for unseparated diffuser flows.

Yungster, Shaye (University of Washington): "Numerical Simulation of Hypervelocity Projectiles in Detonable Gases"

A numerical study of a ramjet-in-tube concept known as the "ram accelerator" is presented. The ram accelerator is a hypervelocity projectile accelerator whose gas-dynamic principles are similar to those of an airbreathing ramjet, but operates in a different manner. The projectile resembles the centerbody of a ramjet and travels through a stationary tube filled with a premixed gaseous fuel and oxidizer mixture. The tube acts as the outer cowling of the ramjet, and the combustion process travels with the projectile, generating a pressure field which produces forward thrust on the projectile. Although several ram accelerator operation modes have been proposed, the discussion will center on the "superdetonative operation mode" (i.e., operating at speeds above the Chapman-Jouguet detonation speed of the mixture). The flow and combustion processes in this ram accelerator operation mode are very similar to the supersonic combustion flow fields of interest to the NASP program.

The numerical analysis of the ram accelerator concept is carried out using a TVD scheme that includes nonequilibrium chemistry, real gas effects, and a 7 species-8 reaction combustion model for hydrogen/oxygen mixtures. Inviscid flow is assumed. The flow, combustion phenomena, and performance characteristics of the ram accelerator are investigated for several projectile configurations in the superdetonative velocity range of 5.0 to 10.0 km/s. The results show that efficient acceleration of projectiles is possible through velocities as high as 9 km/sec. Several code validation tests conducted on blunt projectiles fired into detonable gases are also presented, and the numerical results are compared with experimental results.

CENTER FOR MODELING OF TURBULENCE AND TRANSITION

The Center for Modeling of Turbulence and Transition (CMOTT), a cooperative turbulence research team, was formally organized in June 1990 under the support of Internal Fluid Mechanics Division (IFMD) of NASA Lewis Research Center and the Institute of Computational Mechanics for Propulsion (ICOMP). Dr. Louis Povinelli is the manager of the CMOTT, while Dr. Meng-Sing Liou and Dr. Tsan-Hsing Shih are the coordinator and the technical leader respectively. An advisory group for the CMOTT has also been formed consisting of Professor J.L.Lumley (Cornell University), Dr. M. Goldstein (NASA/LeRC) and Professor E. Reshotko (CWRU). Currently, the CMOTT has eight formal members working on various aspects of turbulence and transition modeling.

The main objective of the CMOTT is to develop, validate, and implement models for turbulence and transition. The flows of interest are 3D flows, incompressible and compressible flows, internal and external flow, and flows with chemical reaction.

The model schemes being studied include the following:

Two-equation (e.g. k-ϵ) and algebraic Reynolds-stress models

Full Reynolds-stress (or second moment closure) models

Probability density function (pdf) models (for chemical reaction)

Renormalization Group Theory (RNG) method

Large eddy simulation (LES) and direct numerical simulation

During the last six months, the CMOTT researchers have presented 9 papers in various conferences including the International Symposium of Engineering Turbulence Modeling and Measurements, NASP Symposium, Lumley's 60th Birthday Symposium, APS, and AIAA (Reno) annual meetings. In addition, five new papers have already been accepted by conferences in 1991 and two others have been submitted to the 1991 Turbulent Shear Flow Conference.

The CMOTT organized a biweekly seminar series in early July 1990. Most speakers were from various branches of IFMD and some from other institutions. Among others it included E. Rice, R. Claus, K. Zaman, J. L. Lumley, M.M.Gibson, and W. Rodi. In addition, the CMOTT has started another biweekly internal group meeting for discussing and exchanging new ideas on modeling and transition.

MORKOVIN LECTURE SERIES

A very successful, intensive two-week long lecture series by Mark V. Morkovin, Professor Emeritus at the Illinois Institute of Technology, was held from April 30 to May 11, 1990. The series was entitled "Instabilities and Transition to Turbulence in Free and Wall Shear Layers." The lecture topics were as follows:

The Nature of Unstable Flow Developments and Turbulence

Illustrations of Viscous, Inviscid, Cross-Flow and Goertler Instabilities, Including Saturation and Breakdown

Successive Instabilities in Mixing Layers, Jets, and Wakes

Primary and Secondary Mechanisms of Wall Layer Instabilities

Effects of Roughness and Introduction to Receptivity

Instabilities, Transition, and Turbulence at Supersonic Speeds

On Bifurcations in Closed and Open Flow Systems with Fixed Dimensions: Rotating Cylindrical Annulus, Ducts and Pipe Flows

Receptivities to Free-Stream Disturbances Including Turbulence and Sound

Phenomena in Bluff Body Wakes

Comments on Chaos: Implications for Philosophy of Research and Design (e-to-the-N). Free-for all discussion

In addition to the more formal lectures, five special informal tutorial sessions were held. Topics for these tutorials were:

Convective, Absolute, and Global Instabilities — Evidence and Implications

Effects of Roughness on Transition

On the Physics of Vorticity, Instabilities, and Turbulence at Supersonic Speeds

Other Bypass Transition — Threshold Disturbances

External Disturbances and Receptivity

The average attendance at the lecture series was 41 while that of the tutorials was 12. Eighty five copies of the lecture transparencies were distributed. There was a strong consensus among those attending that this was a profitable and informative two weeks.
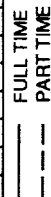
Figure 1. - ICOMP visiting researchers, 1990.

NAME - INSTITUTION		
Atolabi, Dare Purdue U.	S	
Aggarwal, Suresh U. of Illinois-Chicago	F	
Ahn, Kyung Cleveland State	F	
Akl, Fred Louisiana Tech.	S	
Arnone, Andrea U. of Florence	F	
Balsa, Thomas U of Arizona	F	
Bayliss, Alvin Northwestern U.	F	
Bodonyi, Richard Ohio State U.	F	
Braaten, Mark G. E. R&DC	F	
Chang, C. L. Cleveland State U	F	
Chitsomboon, Tawit Old Dominion U.	F	
Choi, Seo-Won Stanford U.	F	
Cowley, Steven Imperial Col.	F	
Deissler, Robert Los Alamos Nat. Lab.	F	
Demuren, Ayodeji* U. of Lagos	F	
Eiseman, Peter Columbia U.	F	
Gajjar, Jitesh Exeter U.	F	
Hagstrom, Thomas‡ SUNY-StonyBrook	F	
Hall, Charles U. of Pittsburgh	F	
Hariharan, S.I. Akron U.	F	
Janus, J. Mark Miss. St. U.	F	
Jiang, Bo-nan U. of Tx-Austin	F	
Kim, Sang-Wook U. of Tx-Arlington	F	

*NOW AT OLD DOMINION U.
‡NOW AT U.NEW MEXICO

S - STRUCTURES
F - FLUID MECHANICS

———— FULL TIME
– – – PART TIME

NAME - INSTITUTION	
Leonard, B. P. Akron U.	F
Lin, Avi Temple U.	F
Liou, William Penn State U.	F
Mathew, Joseph MIT	F
Mawid, Mohammd U. of Illinois-Chicago	F
Messiter, Arthur U. of Michigan	F
Michelassi, Vittorio U. of Florence	F
Mital, Subodh CWRU	S
Pierre, Christophe U. of Michigan	S
Pletcher, Richard Iowa State U.	F
Porsching, Thomas U. of Pittsburgh	F
Shih, Tsan-Hsing Stanford U. CTR	F
Sidi, Avram Technion	F
Steger, Joseph U.of California-Davis	F
Stewart, Mark Princeton U.	F
Swafford, Timothy Miss. St. U.	F
Turkel, Eli Tel Aviv U.	F
Usab, William Purdue U.	F
van Leer, Bram U. of Michigan	F
Whitfield, David Miss. St. U.	F
Wolfshtein, Micha Technion	F
Wong, Yau Shu U. of Alberta	F
Yang, Zhigang Cornell U	F
Yungster, Shaye U. of Washington	F

S - STRUCTURES
F - FLUID MECHANICS

FULL TIME ————
PART TIME — — —

Figure 1. - Concluded.

45

Figure 2. - ICOMP Steering Committee and visiting researchers in August 1990. Kneeling (left to right): Shaye Yungster, Andrea Arnone, Fred Akl, Sang-Wook Kim. 1st row: Subodh Mital, Isaac Greber, Michael Salkind (President, Ohio Aerospace Institute), Louis Povinelli, Dare Aflolabi, Larry Ross (Director, NASA Lewis Research Center), Kyung Ahn, S. I. Hariharan, Arthur Messiter, Marvin Goldstein, Glenda Gamble, Mohammd Mawid. 2nd row: Robert Mullen, C. L. Chang, Avi Lin, Ayodeji Demuren, Charles Feiler, Vittorio Michelassi, Avram Sidi, Lester Nichols, Christophe Pierre, Gregory Swartwout, Eli Reshotko, Stephen Cowley.

UNIVERSITY OR INSTITUTION	NUMBER
1 ALBERTA	1
2 AKRON	2
3 ARIZONA	1
4 CALIFORNIA-DAVIS	1
5 CASE WESTERN RESERVE	1
6 CLEVELAND STATE	2
7 COLUMBIA	1
8 CORNELL	1
9 EXETER	1
10 FLORENCE	2
11 GE R&DC	1
12 ILLINOIS-CHICAGO	2
13 IMPERIAL COLLEGE	1
14 IOWA STATE	1
15 LAGOS	1
16 LOS ALAMOS NATIONAL LAB	1
17 LOUISIANA TECH	1
18 M. I. T.	1
19 MICHIGAN	3
20 MISSISSIPPI STATE	3
21 NORTHWESTERN	1
22 OHIO STATE	1
23 OLD DOMINION	1
24 PENN STATE	1
25 PITTSBURGH	2
26 PRINCETON	1
27 PURDUE	1
28 PURDUE-INDIANAPOLIS	1
29 STANFORD	2
30 SUNY-STONY BROOK	1
31 TECHNION-ISRAEL INSTITUTE OF TECH.	2
32 TEL-AVIV	1
33 TEMPLE	1
34 TEXAS-ARLINGTON	1
35 TEXAS-AUSTIN	1
36 WASHINGTON	1
	47

Figure 3. - Composition of 1990 ICOMP staff - organizations represented.

	1986	1987	1988	1989	1990
PEOPLE	23	43	50	46	47
SEMINARS	10	27	39	30	37
REPORTS	2	9	22	32	25
WORKSHOPS/ LECTURE SERIES PRESENTATIONS	1 7	0 0	2 21	1 14	1 15
UNIVERSITIES OR ORGANIZATIONS	20	28	43	35	36

Figure 4. - ICOMP statistics (1986 to 1990).

Report Documentation Page

1. Report No. NASA TM-103790 ICOMP-91-01	2. Government Accession No.	3. Recipient's Catalog No.

4. Title and Subtitle	5. Report Date
Institute for Computational Mechanics in Propulsion (ICOMP) Fifth Annual Report—1990	May 1991
	6. Performing Organization Code

7. Author(s)	8. Performing Organization Report No.
	E-6067
	10. Work Unit No. 505-62-21

9. Performing Organization Name and Address	11. Contract or Grant No.
National Aeronautics and Space Administration Lewis Research Center Cleveland, Ohio 44135-3191	
	13. Type of Report and Period Covered Technical Memorandum

12. Sponsoring Agency Name and Address	
National Aeronautics and Space Administration Washington, D.C. 20546-0001	14. Sponsoring Agency Code

15. Supplementary Notes

16. Abstract

The Institute for Computational Mechanics in Propulsion (ICOMP) is operated jointly by Case Western Reserve University and the NASA Lewis Research Center in Cleveland, Ohio. The purpose of ICOMP is to develop techniques to improve problem-solving capabilities in all aspects of computational mechanics related to propulsion. This report describes the activities at ICOMP during 1990.

17. Key Words (Suggested by Author(s))	18. Distribution Statement
Numerical analysis Computer science Mathematics	Unclassified - Unlimited Subject Category 64

19. Security Classif. (of the report)	20. Security Classif. (of this page)	21. No. of pages	22. Price*
Unclassified	Unclassified	48	A03